Plasma CNC Owner's Manual

LAGUNA

Laguna Tools
744 Refuge Way
Grand Prairie, TX
lagunatools.com

Scope of This Manual

This manual outlines the basic procedures for the Laguna Tools Plasma CNC machine.

For detailed instructions and videos, please visit www.lagunatools.com.

Customer Service

For technical support, please contact Laguna Tools contact Customer Service by phone at 1-800-332-4094, or email customer_service@lagunatools.com. Please note the machine type in the subject line.

Revision 01 (08/03/2023)

Table of Contents

NOTES:

1.0 General Information and Safety

1.1 Overview

Please read and understand all warnings and operation instructions before using any tool or equipment. Always follow basic safety precautions to reduce the risk of personal injury. Improper operation, maintenance, or modification of tools or equipment could result in serious injury or property damage. Laguna Tools equipment is designed for specific and limited applications. This product should neither be modified nor used for any application other than those for which it was designed.

1.2 Safety Signs and Call-Outs

The purpose of safety signs and call-outs is to draw attention to potential hazards during the operation of the Laguna Tools Plasma. This Owner's Manual makes use of the following call-outs to describe the level of importance of each safety message:

⚠ DANGER

An imminently hazardous situation which, if not avoided, will result in death or serious injury.

⚠ WARNING

A potentially hazardous situation which, if not avoided, may result in death or serious injury.

⚠ CAUTION

A potentially hazardous situation which, if not avoided, could result in minor or moderate injury.

NOTE

A helpful tip from Laguna Tools technical staff.

1.3 Safety Warnings

Failure to comply with safety instructions may lead to personal injury and/or damage to the equipment. Do not operate the machine unless familiar with all safety instructions, warnings, and signs.

1. The machine must be properly electrically grounded. The power supply must be connected with a permanently fixed electrical wire.
2. Keep children and non-operators away from the machine.
3. Operators must be familiar with the installation, operation, and service of the machine. Only proper operation can ensure the safe and smooth running of the machine.

⚠ WARNING

Automated machinery involves moving parts which pose a potential hazard to personnel. Be aware of machine movement at all times.

⚠ WARNING

Only allow qualified service personnel to do electrical installation or repair work, and always disconnect power before accessing or exposing electrical equipment to reduce risks.

⚠ CAUTION

Machine bits are sharp and pose a cutting hazard. Do not handle without gloves or while machine is in operation.

1.4 Additional Safety Information

1. All motion parameters have been set up by Laguna Tools. If any modifications are required, please have a professional operator perform the changes.
2. Safety Signs should be attached to places that are easy to spot.
3. Use the machine only in clean areas free from excessive moisture or flammable objects.
4. The machine must be level. Level the machine if the ground is uneven.
5. Keep the machine and surrounding area clear of obstructions and free from excessive moisture.
6. Keep the machine and cables away from excessive heat, flammable substances, and sharp objects.
7. Do not attempt to exceed the limits of the machine.
8. Disconnect power to all system components when not in use, when changing accessories, and before servicing. Remove the switch keys or lock-out the machine to prevent unauthorized use and child-proof the workshop.
9. Exercise care with machine controls and around keypad to avoid unintentional start-up.
10. Lubricate and change accessories when necessary.
11. Cables and cords should be inspected regularly.
12. Keep controls clean and dry.
13. Keep a copy of this manual for future reference.
14. Perform daily inspection of the machine for damaged, loose, or improperly adjusted parts or any condition that could affect safe operation. For your own safety, do not operate the machine with damaged parts.
15. Stay alert at all times while operating the machine.
16. Always wear safety glasses and hearing protection.
17. Know where the emergency stop switch is located.
18. Never operate machinery under the influence of drugs or alcohol, when tired, or when distracted.
19. Do not wear clothing, apparel, or jewelry that can become entangled in moving parts. Always tie back or cover long hair. Wear non-slip footwear to reduce the risk of slipping and losing control or accidentally contacting cutting tool or moving parts.

20. Never stand on the machine. Serious injury may occur if the machine is tipped or if the cutting tool is unintentionally contacted.
21. Consult the Owner's Manual or Laguna Tools for recommended accessories. Using improper accessories will increase the risk of serious injury or damage.

1.5 Maintenance Personnel

Only qualified personnel should make repairs to this equipment. Use caution and follow procedures when working on the machine. 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 procedure applicable to 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. Read the vendor components manuals for any additional preventative maintenance.
7. Always replace safety guards and other safety devices removed for service and verify 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.

1.6 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. Aways 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 to the machine.

1. Be familiar with the machine. Read and review the manuals carefully.
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 button or the Cycle Stop button function to stop machine motion.

   Never use the machine Dwell Time Code for parts removal or other operator intervention activities that puts the operator in a hazardous position.
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). In case of personal injury notify the Service Department giving a brief description and the date the reported injury occurred. Never operate the machine with someone within a hazardous area.

    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.

1.7 Personal Protection

- 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.

1.8 Eye Protection

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 the cutting area be prepared in such a way as to minimize the reflection and transmission of ultra-violet 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 ultra-violet transmission.

1.9 Safety Instructions for Use

Read the Safety Information in Section 1.0 and the sections of vendor components manuals on safety before beginning to operate the Laguna Tools Plasma Table.

1.10 Input Connections

1. 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.
2. Three-phase input conductors must be sized to carry the rated current of the plasma arc power supply.
3. Primary power cable must be provided with a minimum 600v rating.

1.11 Input Power

1. Connect the ground lead of the four-conductor/three-phase input cable to the electrical system ground in the disconnect box and the ground stud provided in the plasma arc supply.
2. Verify all ground lugs are of adequate size to carry the rated current load.
3. Verify all connections tight to avoid resistance heating.

1.12 Output Power

1. Connect all positive output ground leads to the material grid of the worktable.
2. Connect the material grid of the worktable to a good earth ground.

1.13 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.

⚠ WARNING

The operator and all personnel working in the vicinity of the arc must wear proper protective clothing and equipment.

1.14 Toxic Fumes

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 trichlorethylene will decompose under ultraviolet radiation to form phosgene and other gases. 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 plasm 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.

1.15 Air Contamination

The plasma cutting process generates large quantities of hot metal dust and fumes that would be hazardous if uncontrolled.

A blower pulls a vacuum through the fume extraction assembly in the bed of the machine. The blower pulls the dust-laden air through a customer supplied filter before exhausting the air to the environment.

The gases listed either are produced normally during plasma arc cutting or can form under certain conditions.

1.16 Ozone

Ozone is produced by the reaction of the plasm 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.

1.17 Nitrogen Oxide

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.

1.18 Acetyl Chloride

Acetyl chloride gases from 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 "sweetish" 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 are vapor degreasers containing 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 system:

1. Trichloroethylene
2. Trichloroethane
3. Perchloroethylene
4. Per-Chloroethane
5. Trifluoro-Trichloroethane (Flurocarbon-113)

These chemicals also decompose into small amounts of the toxic gas' phosgene and chorine. You will notice 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 rely on the machine's internal ventilation system to control solvent vapors and their products.

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.

1.19 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 by some vaporized metals from 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 venting to the outside and the machine's internal ventilation system is functioning normally, there should be adequate control of metal fumes during torch cutting.

1.20 Metal Dust

Metal dust is formed as metal vaporizes during torch cutting. A hazard may exist when uncontrolled, excessive levels of metal or dust are produced. If there is proper venting to the outside and the machine's ventilation system is functioning normally, there should be adequate control of metal dust during torch cutting. For proper ventilation, at least 90% of the worktable should be covered by the workpiece (or other sheet metal covers).

See Dust Collect Vendor Installation and Operation Manual for additional precautions.

⚠ WARNING

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

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, 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. These 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 (such as petrol/gasoline).

1.21 Compressed Gas Equipment

Gas cylinders should be mounted securely to a wall or other stable supporting device.

1.22 Cylinders

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. Return these cylinders to the supplier.

1.23 Pressure Regulators

All regulators used to operate plasma equipment must be maintained in a proper working condition. Faulty equipment can cause equipment damage or operator injury. Faulty equipment must be serviced at the manufacturer's 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.

1.24 Hoses

Gas hoses used for Plasma Arc Cutting Systems adhere to the following Color Coding:

- Red – Acetylene
- Orange – LPG
- Blue – 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 kids. 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 standard for more information on hoses.

1.25 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.

⚠ WARNING

Never attempt to operate the plasma unit with any without the power supply covers 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.

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.

1.26 Hot Surfaces

⚠ WARNING

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

⚠ WARNING

During prolonged periods of cutting, parts of the machine bed may become hot to the touch.

⚠ WARNING

Moving machinery can be dangerous.

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.

NOTE

Read this manual thoroughly before operating the machine. Read the Torch Height Control Manual before operating the machine. Read the CNC Control Operator Manual before operating the machine.

1.27 Sparks

Sparks form as the plasma 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 feet (10.7 meters) 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 feet (10.7 meters) of the torch to prevent sparks from passing into adjacent areas.

⚠ WARNING

Sparks from the cutting process may ignite flammable items in the machine bed which may then be drawn into the extraction unit, possibly causing a fire.

NOTE

Use an approved face mask and approved eye protection when cleaning or servicing the dust collector.

NOTE

Plasma arc cutting systems can produce large volumes of fumes. If you exhaust fumes to the outside atmosphere, additional air pollution control devices may be required to conform to local, state, and federal government ordinances. Air pollution control devices are the responsibility of each individual user.

1.28 Internal Ventilation System

Due to the noxious and toxic nature of many torch cutting by-products, Phoenix recommends venting the machine's internal ventilation system (referred to as the dust collector) to the outside. This recommendation is especially important when the shop has one of the following:

1. Low ceilings and/or confirmed area.
2. Large amount of welding and/or Torch Cutting near the Plasma System.
3. Poor Cross Plant Ventilation

1.29 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.

⚠ WARNING

Do not look directly at the arc without proper eye protection.

NOTE

During plasma arc cutting, clothing worn should conform to the instructions presented in the General Safety Requirements section of this manual.

Ultraviolet rays and other radiant energy reflected off the workpiece can produce sunburn. Therefore, when plasma arc cutting is being performed, anyone working withing 25 feet (7.5 meters) of the arc should wear an approved, protective full-face mask, a long-sleeved shirt, gloves, and long pants.

Operations, such as edge cutting, that can cause the arc to be exposed to view should be avoided, because they can increase exposure to radiant energy.

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. Phoenix Plasma recommends that a person with a pacemaker who works near where the 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.

NOTE

There is no history of problems caused to pacemakers by the plasma arc cutting equipment that have been reported to Phoenix Plasma.

1.30 Heat

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, cutter bars, ducting, and dust collector become hot during torch cutting. Avoid contact with these components unless you are wearing heat-resistant gloves.

1.31 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. Phoenix Plasma 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.

⚠ WARNING

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.

NOTE

Noise levels that can cause discomfort or damage to hearing will vary greatly from one individual to another. Phoenix Plasma recommends that ear protection be furnished to any worker who requests it, regardless of applicable industrial standards or tested noise levels.

1.32 Additional Safety Information

The general safety information presented in this chapter does not constitute a complete list of safety instructions for any configuration of the Phoenix Plasma Cutting Table. Warnings and other safety information related to operations described in this manual are presented in the chapters in which those operations are explained. Specific equipment being used by the customer and its application in the customer's factory may require supplementary safety information.

NOTE

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

NOTES:

2.0 Machine Installation

2.1 Receiving the Machine

Following delivery and before the driver and riggers have left, inspect the packing, invoice, and shipping documents. Next, ensure there is no visible damage to the packaging or the machine. All damage must be noted on the delivery documents and signed by the receiver and the delivery driver. Contact Laguna Tools Customer Service as soon as possible in case of damage. It is advisable to photograph and document any shipping damage. The original packaging is required to return damaged equipment to Laguna Tools.

NOTE

Laguna Tools tests all machines prior to shipping, but some adjustments may have to be undertaken by the customer.

Most large machinery will be delivered on a tractor trailer 48 to 53 feet long. Please notify a Sales Representative with any Delivery Restrictions. The customer is required to have a forklift (6000 lbs. or larger is recommended) with 72-inch tynes or tyne extensions.

2.2 Installation

NOTE

In order for the installation of the Plasma CNC machine to be efficient and cost effective, the following tasks must be completed prior to the arrival of a Laguna Tools technician.

Figure 2-1: Power Supply Front

Figure 2-2: Power Supply Rear

1. Automated Cable
2. Compressed Air Cable
3. Power Cable
4. Torch Cable
5. Work Lead

Perform the following steps to prepare the Plasma machine for set-up and training by a Laguna Tools technician.

1. Remove all protective coating and packaging.
2. Verify all items accounted for (refer to the machine Specifications).
3. Before proceeding, verify the workshop has the necessary electrical voltage and amperage to power the machine.
4. Verify the machine is level.
5. Connect the Power Supply.

6. Connect the Automation Cable.
7. Connect the ground wire to the rear of the plasma and to the ground rod (refer to Sections 5.6 – 5.9 for further information).
8. Attach compressed air to the machine. Ensure the compressed air is clean and dry.
9. Prepare an adequate supply of materials for test cuts.
10. Ensure the operator of the Plasma CNC has read the entire Operator's Manual prior to training by the Laguna Tools technician.
11. Ensure the operator has been trained on the FlashCut software used by the Plasma CNC machine.
12. If an autocut was purchased, ensure appropriate gases are available for the material that will be cut.

Should a Laguna Tools technician not be able to come to the site and work on the machines because the procedures listed above have not been followed, any additional expense incurred will be passed on to the purchaser of the machine.

If any of the above procedures cannot be completed, the purchaser will cover all expenses incurred during the setup and training period. This includes, but is not limited to, airfare, rental car, hotel, parking, and/or toll fees per diem plus $115.50 per hour.

NOTES:

3.0 Plasma Overview

The Laguna Plasma combines a heavy all-steel frame, FlashCut controls, and servos running on helical rack and pinion. It can easily handle high production loads without sacrificing accuracy or cut quantity. The Laguna Tools Plasma is a rugged platform with the durability and accuracy to process materials including wood, plastics, foams, aluminum, and composites.

3.1 Components

1. **Bed** – The bed of the machine consists of a heavy steel frame with steel supporting plates. The supporting plates are designed to give point contact with the job. The plates will be cut by the plasma cutting head and are replaceable.
2. **Gantry** – The gantry straddles the bed and carries the plasma cutting head motion system. It is moved along the length of the bed by a precision rack-and-pinion system that is controlled by the machine controller.
3. **Plasma Cutting Head** – The plasma cutting head is moved along the gantry by a precision rack-and-pinion system that is controlled by the machine controller. The plasma cutting head is moved vertically by a precision ball screw system that is controlled by the machine controller.
4. **Frame** – The frame is a heavy welded construction that supports all the other parts of the machine.
5. **Controller Stand** – The controller stand provides a sturdy platform for the monitor, keyboard, and CPU.
6. **Caterpillar Track** – The caterpillar track runs along the side of the machine and across the gantry in a trough and carries all the electrical cables and gas pipes.
7. **Oiler** – The oiler connects to all the relevant slides on the machine and when pumped by hand will lubricate all the relevant slides.
8. **Supporting Feet** – There are supporting feet that are used to level the machine.

3.2 Supplied with the Machine

The items supplied with the machine will depend on the specification of the particular machine and any ordered extras.

3.3 Specifications

Machine	Level 1 – C404	Level 2 – C408	Level 3 – C510
X-Axis Travel	49.0 inches	49.0 inches	61.0 inches
Y-Axis Travel	49.0 inches	97.0 inches	121.0 inches
Approximate Dimensions (In.)	82L x 92W x 60H	138L x 92W x 60H	158L x 102W x 60H
Approximate Weight (Lbs.)	1500	2000	2500
Maximum Rapid Speed	393 Inches/min.		
CNC Control	Starfire		
Frame Construction	Welded Tubular Steel		
Bridge Construction	Welded Tubular Steel		
Drive Motors	X-Axis: Stepper Motor Y-Axis: (2) Two Stepper Motors Torch Height Control: Stepper Motor		
Mechanical Drive System	X-Axis: Rack & Pinion (2) Y-Axis: Rack & Pinion Z-Axis: Ball Screw		
Available Plasma Torches	Hypertherm Powermax 45XP Hypertherm Powermax 65 Hypertherm Powermax 85		
Torch Control	Arc Voltage with Initial Height Sensing		
Torch Protection	Pneumatic Breakaway Design		
Exhaust Fan	12000 cfm @ 3" H2O		
Available Options	Pipe Cutting Attachment Water Mist Chiller		

4.0 Machine Placement

When unpacking the Plasma, separate all enclosed items from the packing materials and inspect each for damage. Save the packaging materials until all issues concerning missing or damaged items have been resolved.

4.1 Placement

Select the area where the Plasma will be operated. The physical environment where the Plasma is located is important to safe assembly and operation. Before removing the Plasma from its packaging consider the weight load, electrical installation requirements, lighting, and space allocation available for the machine and accompanying materials.

Guidelines for properly placing the machine follow:

1. There should be sufficient area around the machine to facilitate easy access to the workpiece, perform maintenance, and provide safe egress in the event of an emergency.
2. Select a solid level floor rated to hold the weight of the Plasma and workpieces under both static and dynamic loads. Laguna Tools recommends concrete flooring. Consult a licensed and experienced professional if in doubt.
3. Locate the Plasma close to a power source.
4. Allow an area for the storage of workpiece materials, finished products, and tools.
5. Leave ample space around the machine for the operation to handle both the equipment and the materials being cut.

4.2 Leveling

Once placed in a designated area, the machine will need to be leveled. Four (4) leveling pucks are located under the table. Using a six (6) foot level, adjust the leveling pucks to level the machine. Once the machine is level, unwrap the shrink-wrapped components.

4.3 Electrical Requirements

The Laguna Plasma requires permanent, direct power installed by a qualified electrician familiar with industrial best practices. Ensure that all power cords are protected from traffic, moisture,

chemicals, or other hazards. For safety, always have a qualified electrician assess grounding and any further electrical needs.

4.4 Lighting

Ensure that the lighting your machine is placed under is sufficient to safely perform regular operation and maintenance. Any glares, shadows, or strobe lighting which may distract or prevent the operator from safely operating the machinery should be removed from the working area.

4.5 Unpacking the Machine

Unpacking the Plasma will require tin snips, a knife, and a wrench.

1. Using the tin snips, cut the banding that is securing the Plasma to the pallet.
2. Ordered parts will be packed on the machine.
3. Professional riggers are required for installation of the Laguna Plasma.
4. Approaching the machine from the long side and lift the machine by the frame taking care that there are no cables or pipes around the forklift tynes. The caterpillar track tray, vacuum hoses, air lines, and cables under the Plasma are very susceptible to pinch and crush damage.
5. Lower the machine gently to the floor.

5.0 Assembly and Setup

5.1 Cleaning the Machine

The machine is shipped with non-painted surfaces protected from rust by a film of grease. The grease must be removed as it will attract dirt. Surfaces should be coated with a 30W oil or wax. Remove any excess.

5.2 Electrical Connection for the Machine

The plug for the main power cable will depend on your installation. Ensure that when installing the electrical supply to the machine that 220V three-phase is supplied. It is not possible to recommend a breaker size, as this will be dependent on the specification of the particular machine.

NOTE

When wiring the machine to the electrical system, keep the cable as short as possible, and the cable should not be allowed to run along the floor as this will cause a trip hazard.

NOTE

A qualified electrician must carry out the electrical installation.

5.3 Shielding and Grounding

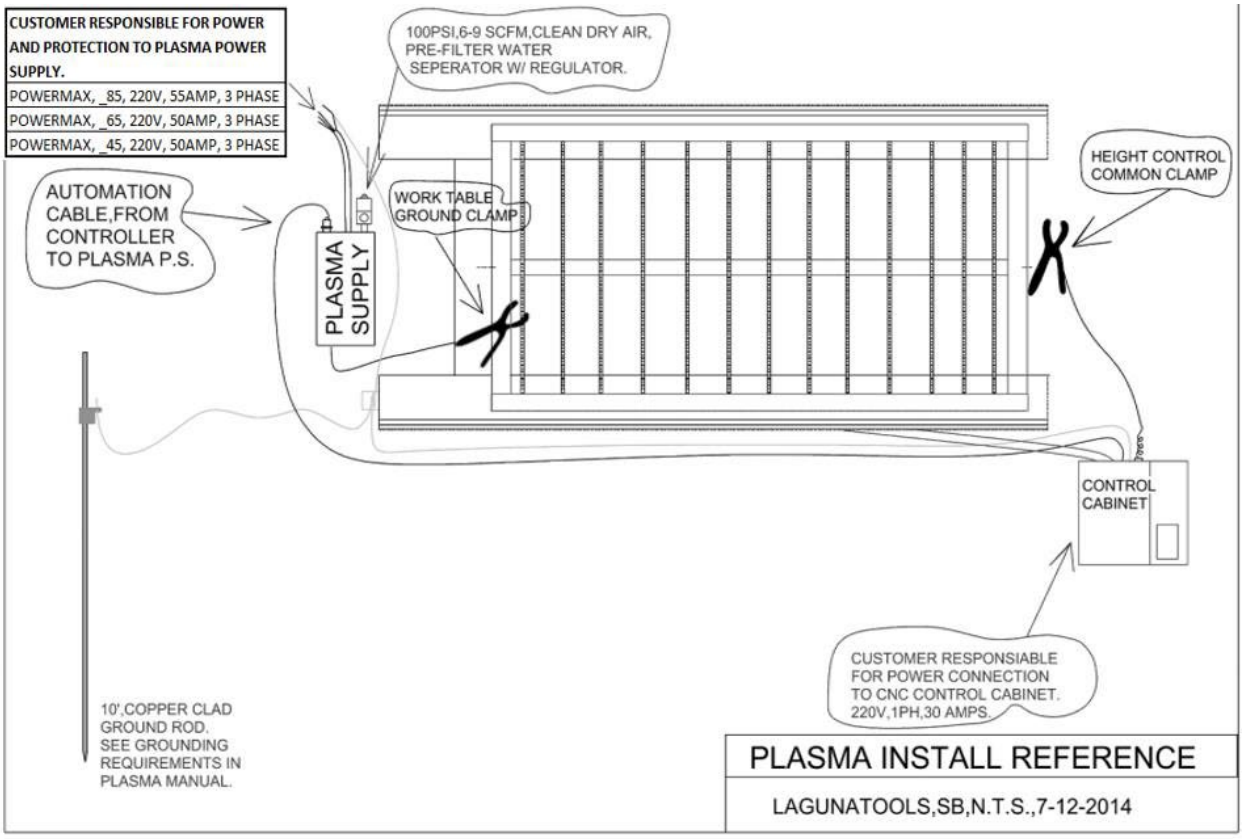
This section is preliminary and offers typical shielding and grounding techniques to reduce RF noise. As with all electrical devices it is important to always follow all local electrical codes including but not limited to the National Electrical Code.

5.4 Motor Controller and Other Controls

Keep the motor controller and other controls as far from the plasma power supply as possible.

5.5 Plasma Power Supply

Place the plasma power supply and/or remote arc starter at the rear of the frame. Keep the CNC computer controller as far from the power supply and/or remote arc starter as possible. Follow the power supply grounding diagram found in the system manual.



5.6 Grounding Connections

Pilot arc starting generates a certain amount of Electromagnetic Interference (EMI). This is commonly called RF noise. The RF noise may interfere with other electronic equipment such as the CNC controller and other equipment in the vicinity. To minimize the RF interference the following grounding procedures should be followed when installation of the machine is undertaken.

Drive 5/8-inch diameter x 10 feet copper clad steel earth rod as close to the table as possible. It may be necessary to weld a second rod to reach the moisture layer to achieve a proper ground.

Follow the attached instructions for proper earth ground rod testing. If multiple ground rods are required to achieve proper grounding keep a minimum distance of 1.1 times the drive length between driven rods.

Create a main star ground on one of the table legs. The main star ground will be the only point where all grounding wires will terminate before the earth ground rod.

The point should have good metal to metal contact and provide enough ground lead attachment points to eliminate the need to stack lead terminals.

NOTE

Keep the ground wires as short as possible. Remove any paint, rust, or oxide from the connection point. Always use star washers and electrical anti-corrosion paste on all connections.

Connect a minimum 1/0 AWG cable (4/0 is best) from the start to the earth ground rod. Do not connect any other grounds directly to the ground rod.

- Create an auxiliary start ground point as needed to connect ground leads from moving points of the table to the main ground. The point should have good metal to metal contact.
- Connect this start to the main star using #4 AWG minimum.
- Connect the power supply safety ground directly to the main start ground with a minimum 1/0 cable.
- Connect the remote arc starter directly to the main start ground with a minimum 1/0 cable.

5.7 Diode and Grounding

The plasma waterbed features a diode at the rear of the table. This reduces electrical feedback to sensitive electronic components. The ground from the plasma power unit should attach to the end

of the diode. The earth ground will attach to a bolt threaded just to the right of the diode. The table will function with the cables reversed or without a ground, but there is a high probability of poor cuts or shortened component lifespan which may not be covered under warranty.

A blow-through table without a water pan will have no diode. In this case, both the work ground and the earth ground are the same bolt.

5.8 Grounding Test

⚠ CAUTION

Failure to carefully follow these instructions will result in erratic torch height behavior.

Install the torch per Hypertherm directions. Attach the torch to the Z head using the attached magnet. Route the cable away from the servo wiring tracks. Cross the wiring tracks at ninety degrees.

Connect the wire to the machine grounding lug and the Hypertherm grounding lug.

Provide a path to the earth ground. This ground can be created by drilling a hole in the floor and installing a 10-foot copper ground rod into the earth. Do not use a stake from the main electrical system. Attach a wire that is at least 10 gauge to the ground lug on the back of the plasma and to the copper rod. A stranded wire is better.

NOTE

Earth ground is not guaranteed by the copper stake in the ground.

Dirt is not by itself conductive. Multiple elements combine to make earth conductive enough to be an effective ground.

Moisture is an important factor in conductivity. As the earth dries, it loses conductivity.

It may be necessary to use multiple ground rods or dig deeper than ten (10) feet to provide an effective ground.

Other means to improve ground include:

1. Pour water down the ground hole.
2. Add salt to the water mixture.
3. Drill a larger hole and add a mixture of sand and salt.

NOTE

Increasing the ground rod length beyond 20-30 feet (6.1 – 9.1 meters) does not generally increase the effectiveness of the ground rod. A larger diameter rod which has more surface area may help.

NOTE

Keep the torch lead cleans. Dirt and Metal particles bleed off energy, which causes difficult starting and increases chances of RF interference.

In order to verify there is an effective earth ground, an electrician should perform either an earth ground test or light bulb test.

The earth ground test requires special equipment and is more expensive than the light bulb test.

⚠ WARNING

Earth ground tests must be performed by a certified electrician. There is a risk of injury or death from electric shock.

The light bulb test requires a 100-watt light bulb in a plastic or porcelain fixture. Apply power on the side of the light bulb fixture. Attach the other side will be attached to the stud on the table where the ground rod is connected. Attach the neutral wire from the same circuit to a voltmeter and the other side of the voltmeter will go to the ground rod. There should be no more than 0.75 volts on the meter. A reading of more than 0.75 volts indicates an ineffective ground (refer to the diagram in Section 5.9).

Ideally, the reading on the multimeter should be 3 VACS for 115VAC or 1.5VAC for 230 VAC line.

Once a proper earth ground is attained, connect a wire from the ground lug on the rear of the machine to the ground stake using the provided clamp.

The system in now ready for start-up.

5.9 Ground Integrity Test Procedure

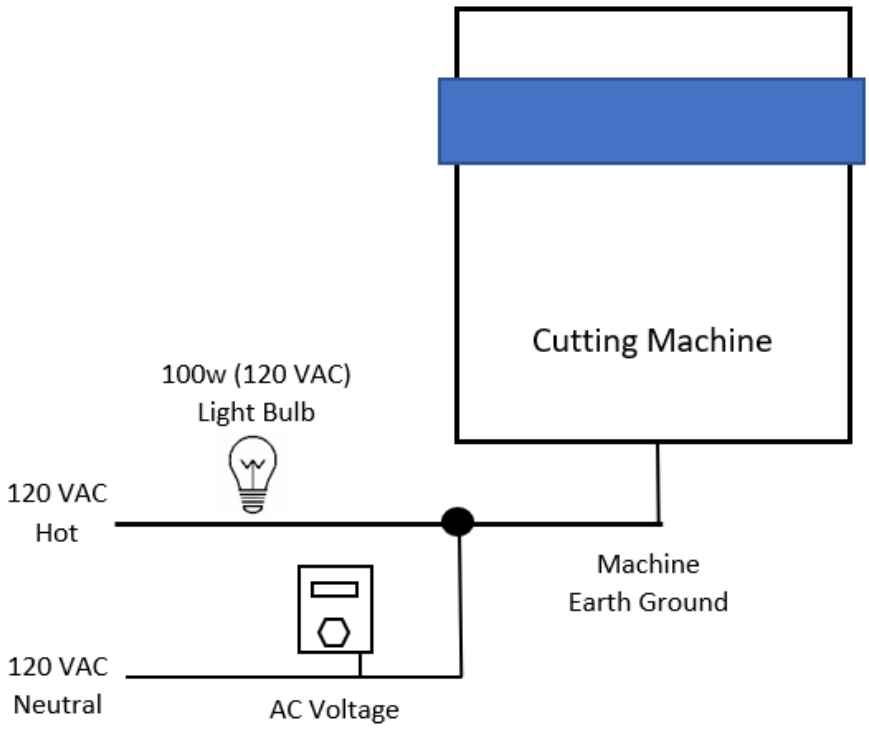
⚠ WARNING

All work must be performed by qualified personnel.

Use the following procedure during the installation of plasma systems or any time a poor ground is suspected.

This test can only performed using 120 VAC with a neutral connected to chassis ground at the power distribution panel.

The cutting machine must be grounded to a good earth using a conductor larger than 10 gauge (mm^2).



5.10 Shielding

5.10.1 Single Shield Method

Connect the shield/drain of a shielded cable to ground at the sensitive equipment end and leave the other end of the shield disconnected. Cover all non-shielded cables with a tightly woven braided shield. Slide the shield over the cable leaving enough to slide back over the electrical connectors and terminate only the end to the sensitive equipment side ground. Use a metal cable clamp to attach the shield to the ground.

5.10.2 Double Shield Method (Preferred)

Cover all shielded cables with a tightly woven Braided Shield. Slide the shield over the cable leaving enough to slide back over the electrical connectors. Use a metal cable clamp to attach the shield to the nearest ground at both ends of the cable.

Clamp on Ferrite Cores – Use a Clamp Type Ferrite Core on the following cables:

- Wall Mount Transformers
- RS232 Cables (as an alternative, fiber optic converters will offer the best noise immunity)
- Computer and Motor Controller AC power cord
- Control cables and encoder cables
- I/O Cables

Open the Ferrites Clamp to Pass 4 to 10 turns around one half of the core then close and snap the clamp shut.

Ferrite Cores are not recommended for use with Braided Shielding.

- If cables must cross ground wires or at worse case torch leads do so at 90-degree angles.
- Select a power outlet rated to support the devices that will be connected.
- Avoid connecting the cutting table to multiple AC power outlets. Use one outlet with a good surge suppression device and multiple power outlets.
- The Plasma Power Supply may be connected to a different power outlet however it is best if the outlet is common to the same panel as the cutting table.
- Do not coil wires and cables. If cleanup of the installation is required, tie the loops of extra cable so the major length of the cable is touching. Remove as much air gap as possible from the cable bunch.

6.0 Basic Machine Operations

6.1 Turning on the Plasma Machine

1. Verify the Emergency Stop button is disengaged or in the "out" position.
2. Turn on the controller.
3. Turn on the Power Supply by turning the switch clockwise.

6.2 Software Installation

The Laguna Plasma machine is operated with FlashCut Software installed on the computer controller. FlashCut Software should be installed before connecting to a signal generator. Refer to the FlashCut 9.0 User's Guide for complete installation instructions.

6.3 Homing the Machine

1. Power on the controller.
2. Agree to the Safety Terms in the pop-up window.
3. Click Yes in the Warning pop-up window.
4. The machine will automatically Home.

6.4 Preparing the Machine

1. Verify the machine is grounded and the power is on.
2. Home the machine.
3. Load the project material onto the bed.
4. Secure the workpiece.
5. Jog the machine to the starting position.
6. Zero the reference origin.

NOTES:

7.0 Introduction to FlashCut CAD/CAM and CNC Control Software

FlashCut Version 9 has three (3) main portions – FlashCut CAD, FlashCut CAM, and FlashCut CNC.

- FlashCut CAD – A simple and powerful 2-D CAD program, with a full-featured drawing environment. Includes support for importing and exporting DXF files.
- FlashCut CAM – An integrated CAM program with many advanced features, including kerf compensation and true shape nesting.
- FlashCut CNC – The latest version of our control software which can be configured to operate almost any type of automated machine tool. Includes full support for editing, importing, and exporting G-Code files.

FlashCut CAD and FlashCut CAM are sold as a combined product. FlashCut CNC is sold separately, but all three (3) together create a single integrated program.

We are committed to the excellence and ongoing enhancement of FlashCut CNC. Feel free to contact us with any comments or questions.

7.1 System Requirements

- Windows Version 7, 8, or 10 (64 Bit)
- Intel I5 processor (or equivalent) or better
- At least 4GB of RAM
- A separate graphics card is preferred for processing larger files

7.2 Program Packages and Features

FlashCut CNC is available in different configurations depending on your needs. The standard package contains the features needed by most customers to design and cut 2-D parts. Pro and premium packages offer advanced functionality and enhanced machine control during production. Some specific advanced features can be purchased a la carte. See the Version 9.0 Feature Summary for more information.

7.3 Version Comparison

The table below lists the different features included in the standard, professional, and premium packages, and those available a la carte. Please contact FlashCut for more information when ordering.

Version 9.0 Feature Summary	Standard Package	Pro Package	Premium Package
Simple, powerful 2-D CAD with improved DXF/DWG import	x	x	x
Measuring tool displays any part dimension or distance	x	x	x
Free form stretching and manipulation	x	x	x
Layer support	x	x	x
Centerline image import	x	x	x
Silhouette image import	x	x	x
Integrated CAM with true shape nesting with improved performance	x	x	x
Commonly used plate list	x	x	x
Customizable cut charts	x	x	x
Customizable default lead-in, lead-out strategies	x	x	x
Editable plasma and oxyfuel cut charts for Hypertherm® and Thermal Dynamics®	x	x	x
Oxyfuel support	x	x	x
Plasma support	x	x	x
Automatic kerf compensation	x	x	x
Grid nesting	x	x	x
Graphical editing of lead in and lead out locations	x	x	x
Tabbing	x	x	x
Overburn support	x	x	x
Assign different operations to each part	x	x	x
Cut sequencing	x	x	x

Version 9.0 Feature Summary	Standard Package	Pro Package	Premium Package
Drag, Delete, Rotate, Copy parts on the nest	x	x	x
Operation sequencing	x	x	x
Cutout or Part Mode	x	x	x
Custom views of kerf width, cut direction, toolpath, rapids, and part geometry	x	x	x
Customizable post processor	x	x	x
Easy CNC program flow control including jump to line, forward, and reverse processing	x	x	x
Dry run CNC	x	x	x
Momentary run CNC	x	x	x
Kerf crossing detection	x	x	x
THC performance history chart	x	x	x
Smart feed rate and Torch Height Control (THC) processing for small holes	o	x	x
Curved text around any shape	o	x	x
DXF export	o	x	x
Marker and plasma marking	o	x	x
True type font support without exploding	o	x	X
Mill router	o	o	x
3D cutting simulation	o	o	x
Laser pointer and laser pointer software support*	o	o	o
Integrated shape library	o	x	x
Fix drawing tool finds and corrects CAD mistakes	o	x	x
Bridge entities tool	o	x	x
Shape welding tool	o	x	x
Part shading	o	x	x
True shape nesting with improved performance	o	x	x

Version 9.0 Feature Summary	Standard Package	Pro Package	Premium Package
Import of parts from external files	o	x	x
Multi-sheet and fill sheet true shape nesting	o	x	x
Corner looping	o	x	x
Automatic Lead-In on Restart	o	x	x
Dynamic THC on/off	o	x	x
Rip cutting	o	x	x
Advanced G-code file management	o	o	x
Dynamic ventilation	o	o	x
Coordinate rotation for workpiece alignment	o	o	x
Rotary axis support	o	o	o
Compound (dual) rotary table support	o	o	o
Programmable I/O interface board*	o	o	o
Joystick with Programmable I/O Interface Board *	o	o	o
Integrated THC with Mini I/O Expansion Board, Voltage Isolation Box, Ohmic and Torch Interface Cables *	o	o	o
RS-485 communications with Hypertherm® Powermax® plasmas for real-time current and pressure control *	o	o	o
Smart touch off	o	o	o
Laser cutter support		o	x
Support for multiple simultaneous fabrication heads		o	x
Part numbers		o	x
Waterjet support		o	x
Multi-layer color image import		x	x
Advanced lead in and lead out editing		x	x
Animated 3D cutting simulation		x	x
Nesting report to estimate cost and weight of materials		o	x

Version 9.0 Feature Summary	Standard Package	Pro Package	Premium Package
CNC progress meter		x	x
Move to point in CNC viewport		x	x
Motor current sensing		o	o
* Includes hardware			

7.4 Installing the FlashCut CNC Software

7.4.1 Preparing to Install the Software

NOTE

Important: Install FlashCut CNC before connecting to a signal generator.

The software should be installed before you connect a signal generator to the PC. If you connect the electronic hardware before installing the software, you may have to manually install the USB driver. Instructions for **Installing the USB Driver** are provided below.

When the hardware is connected to the computer with the USB cable and then powered on, Windows will try and find a driver for the device. If the correct driver is not already installed, Windows may attempt to use a driver for a different device. If the software is installed first, the proper driver will be selected.

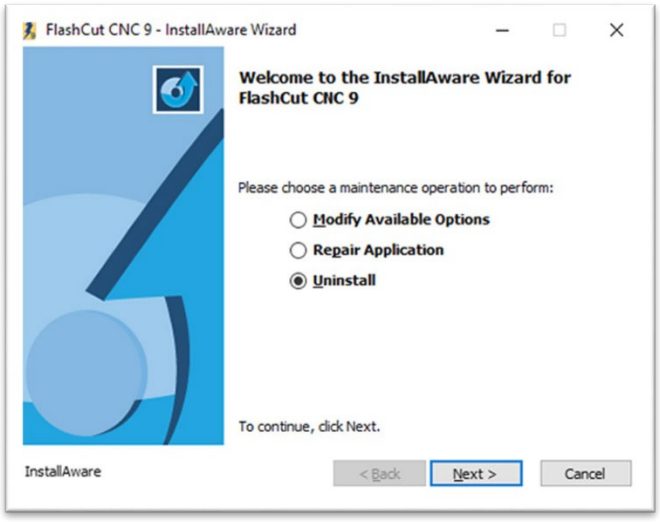
7.4.2 Installing the Software

Please follow these instructions to install FlashCut CNC.

1. Open the FlashCut CNC installer downloaded from the internet or mount the installation CD.
2. When prompted to run the installer, select Run to open the Install Aware wizard.
3. The wizard checks for previous installations of FlashCut CNC version 5 through 8 and prompts you to uninstall them.

NOTE

You may also uninstall a version of FlashCut CNC 9 manually by selecting Uninstall FlashCut CNC 8 from the Windows Start menu.

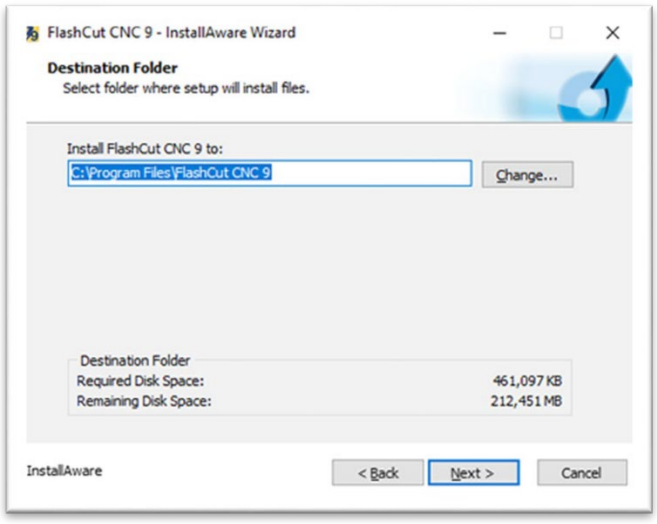


Click **Next >**.

4. After dealing with the pre-installation requirements, the wizard prompts you to install the latest release.
5. Review the software license agreement.

Click **Next >**.

6. Select a destination folder for program files. By default, FlashCut creates a new folder, as shown here. Click **Change**… to set a new location for the file.



Click **Next >**.

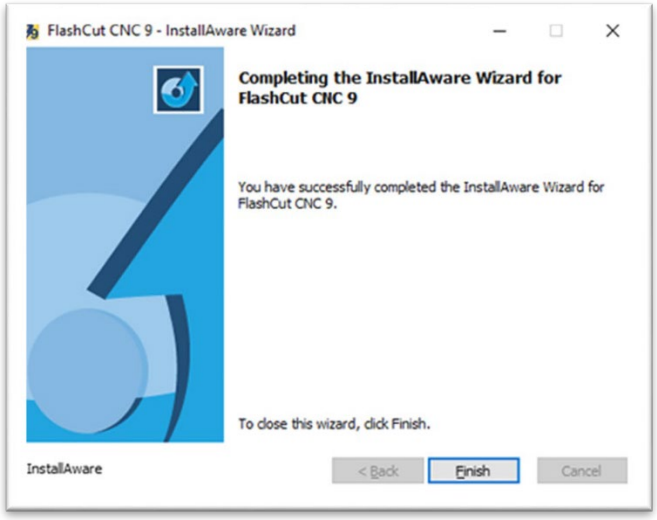
7. The wizard is now ready to perform the installation and configuration.

Click **Next >** to finalize the configuration and begin the installation.

8. During the installation process, Windows may display this message asking for permission to install the USB Signal Generator 501A driver.

   Click **Install** to install the driver software.

9. When complete, the wizard will inform you that the installation was successful.



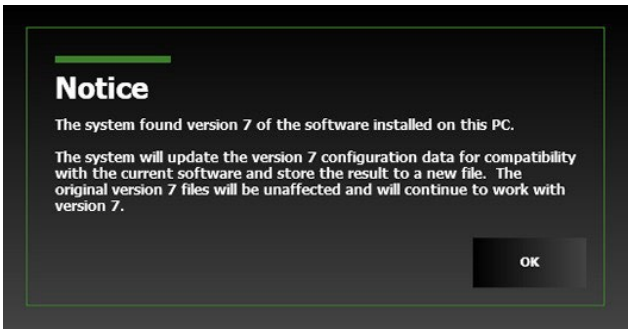
Click **Finish** to close the wizard.

NOTE

The wizard may give you the option to restart your computer by checking the* Restart now *box. The option to restart your computer is typically only offered if FlashCut software has never been installed on this computer before.

7.5 Updating the Setup File

If an older version of the setup file is present, FlashCut will save a copy and update the file when you launch the updated version.



For more information on the setup file, see **Configuring FlashCut**.

7.6 Installing the USB Driver

The software needs to be installed before you connect a signal generator to the PC. If you connect the electronic hardware before installing the software, there is a high likelihood that you will have to manually install the USB driver. The USB driver will be automatically installed as long as the FlashCut CNC software has been successfully installed prior to connecting the controller with the USB cable and powering it on.

If you need to manually install the FlashCut USB driver so the software can communicate with the signal generator, follow these instructions.

1. Make sure the FlashCut software is installed.
2. Connect the signal generator to the PC using an A-B USB cable.
3. Turn on the signal generator or CNC controller.

To finish the driver installation, follow the steps for your version of Windows.

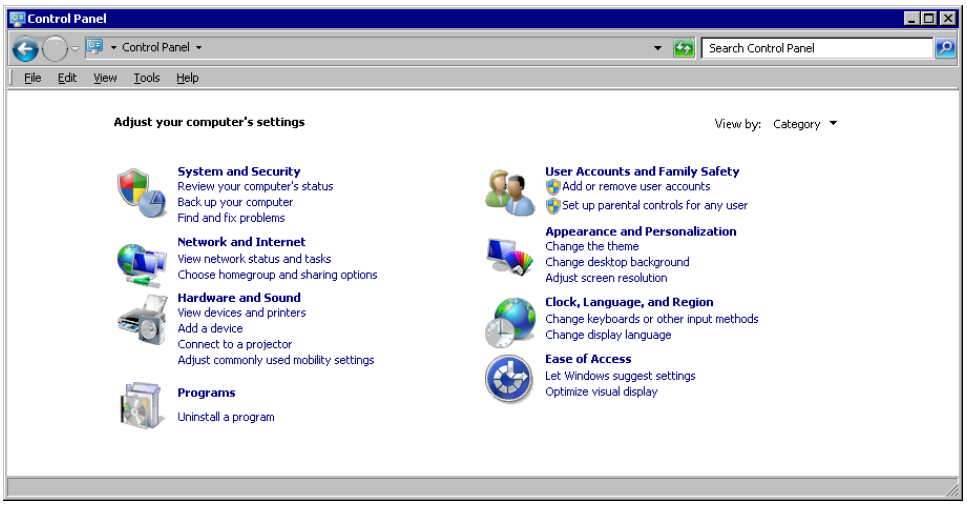
7.6.1 Windows 8 and 10

Later Window versions automatically find and install the driver for the USB signal generator. If Windows is unable to locate and install the drivers, proceed with the following steps in order to properly install the driver.

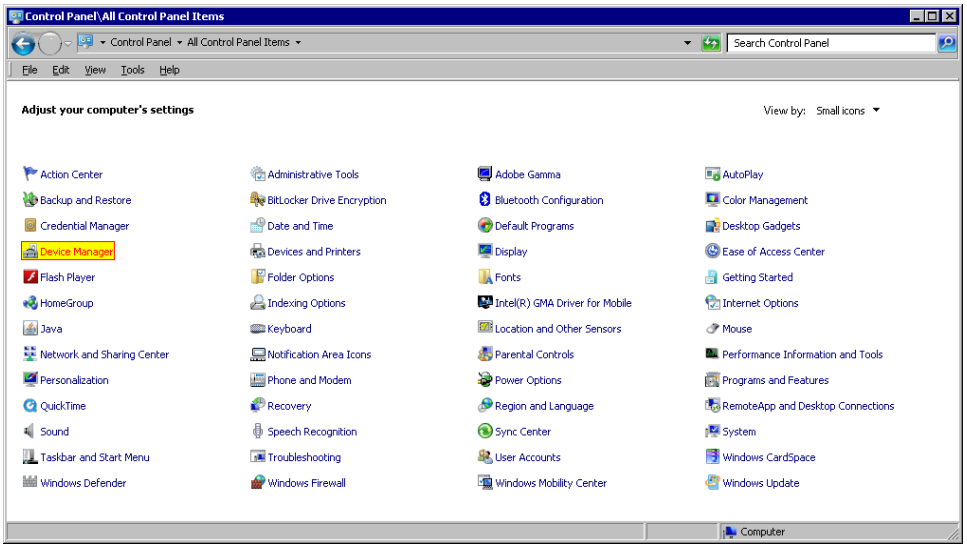
7.6.2 Windows 7

Windows will automatically find and install the driver for the USB signal generator. If Windows is unable to locate and install the drivers, proceed with the following steps in order to properly install the driver.

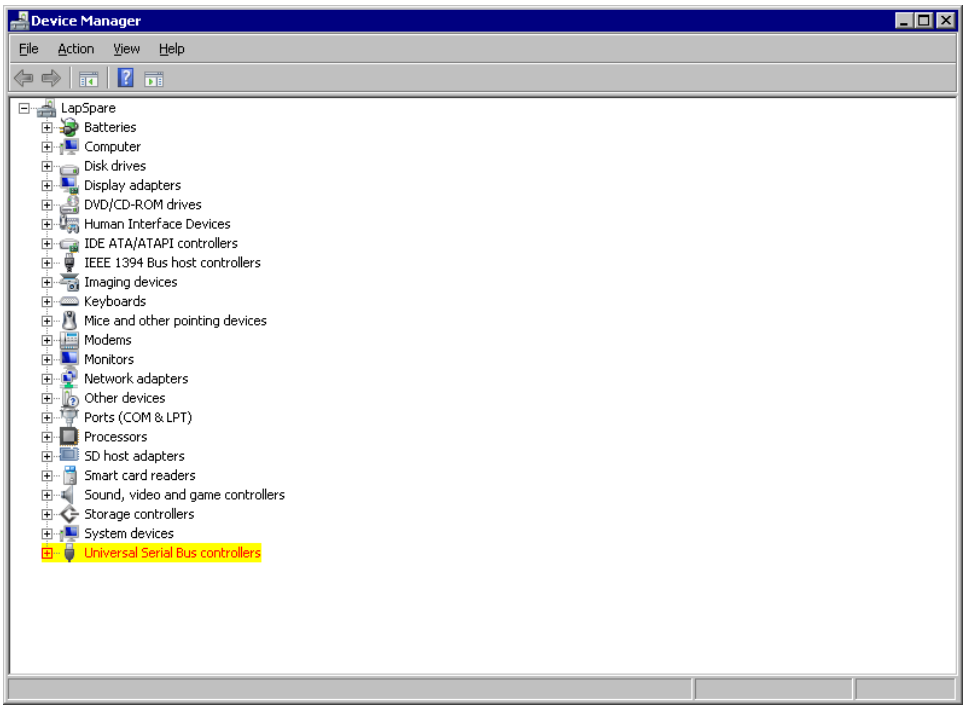
- Click Start►Control Panel. Windows displays the Control Panel screen.



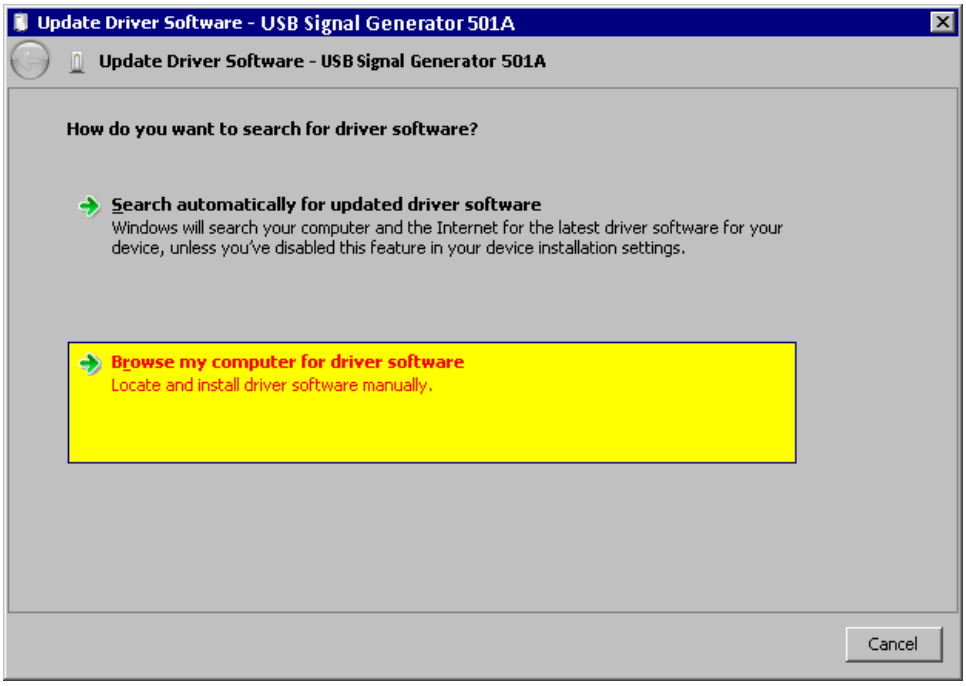
- At the top right corner of the screen, click View By and select Large Icons or Small Icons. Windows display the available Control Panel icons.



- Click Device Manager. Windows displays the Device Manager screen.

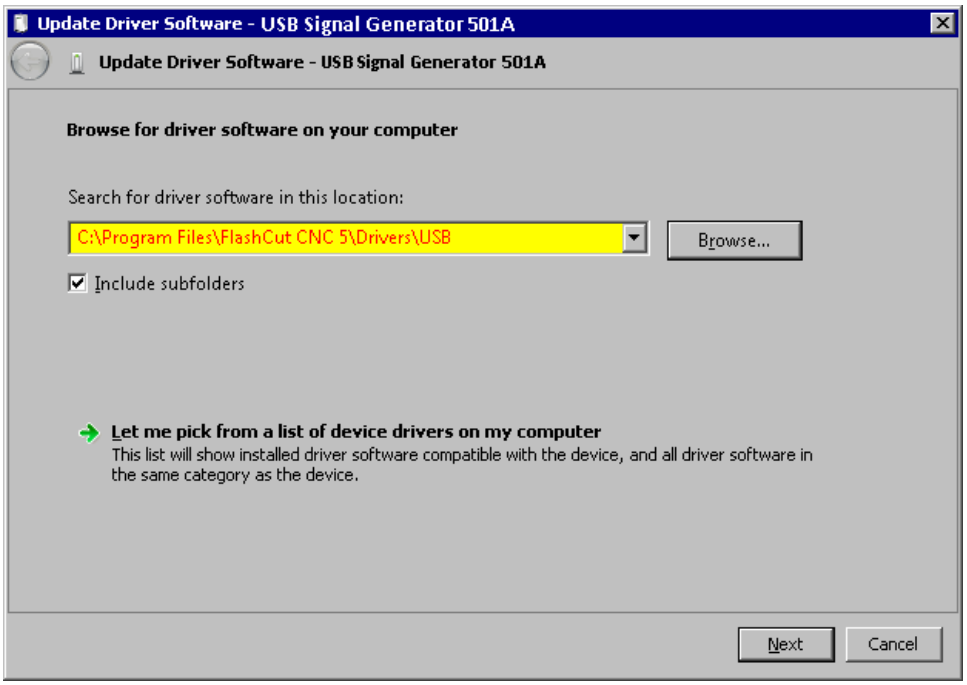


- Expand the Universal Serial Bus Controllers listing.
- Right click on USB Signal Generator 501A and select Update Driver Software. Windows displays this dialog box.



Click **Browse my computer for driver software.**

- Windows displays the following dialog box.

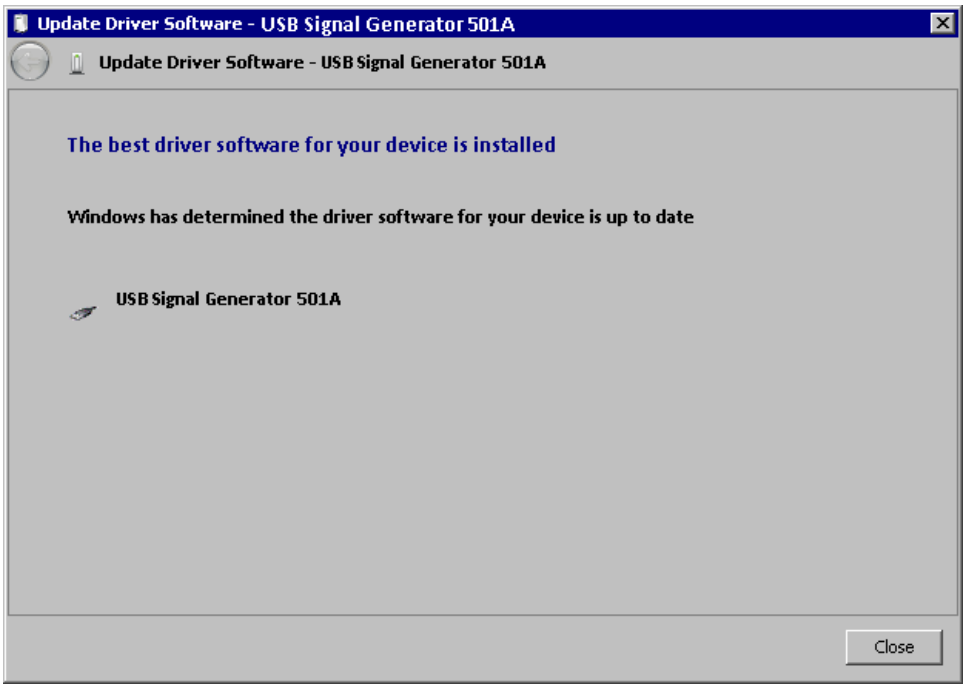


- Click **Browse** and navigate to the following folder:

  C:\Program Files\FlashCut CNC 8\Drivers\USB

When you have navigated to the appropriate folder, click **Next >**.

- Windows installs the driver then displays this message to tell you that the driver installation is complete.

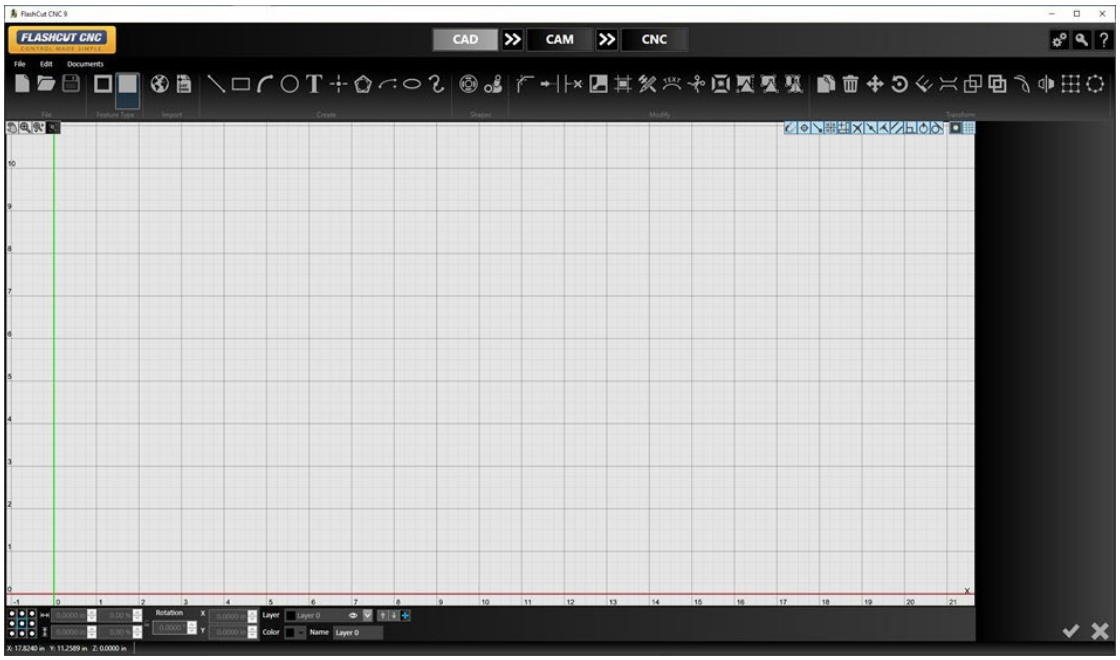


Click **Close.**

7.7 Getting Started with FlashCut

1. Launching the Software

When you start the program, you will see the drawing workspace. From here you can either open an existing file, or simply begin drawing.



2. **Running in Evaluation Mode**

   After you install the program, FlashCut CNC will run in evaluation mode until you activate a license. 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.

3. **Activating a License**

   To enable the full functionality of the program, you must activate a PC license. Select the License button in any workspace. For more information, see **Licensing FlashCut**.

4. **Configuring the Software**

   When you first launch the program, you will be prompted for the location of the FlashCut Data folder. The default location is C:\FlashCut Data\FlashCut CNC 8.

   To examine or change settings saved in this folder, select the Configuration button in any workspace. For more information, see **Configuring FlashCut**.

5. **Getting Help**

   To view the program version, firmware, graphics driver status, select the Help button in any

   workspace. From the Help window, you may also view this user's guide, or the Programming Reference.

   In addition, you can build a Support File to assist in troubleshooting problems. For more information, see **Getting Help**.

7.8 Understanding Common Interface Elements

7.8.1 Configuration, License, and Help Buttons

The Configuration, License, and Help buttons are the same in FlashCut CAD, CAM, and CNC.

The Configuration interface modifies program settings stored in the setup file. for more information.

The License interface is used to install license files and activate or deactivate program licenses. See **Licensing FlashCut** for more information.

The Help interface is used to display version numbers, and to build a support file for use during troubleshooting. See **Getting Help** for more information.

7.8.2 Tabs

Use the tabs at the top of the screen to navigate between the CAD, CAM, and CNC windows.

Use the Double Arrow tabs to send the drawing you are working on to the next stage: from CAD to CAM to create the toolpath, or from CAM to CNC to generate the G-Code and cut the parts.

FlashCut CAD

arrows send to CAM and create toolpath

FlashCut CAM

arrows send toolpath to CNC and generate G-Code

FlashCut CNC

7.8.3 Accept/Cancel

The Accept and Cancel buttons are located in the bottom right of the window in FlashCut CAD and FlashCut CAM.

The green check is used to Accept an action or operation, and the red X is used to Cancel an action or operation. The Parameters area often displays detailed information about what you can Accept or Cancel.

7.8.4 Menu Bar

The menu bar is the same in both FlashCut CAD and FlashCut CAM. The menu bar contains menu items for system commands. Hotkey commands are listed next to their respective functions. Accelerator keys are displayed when you press the {Alt} key.

- File menu
- Edit menu
- Documents menu

7.8.5 File Menu

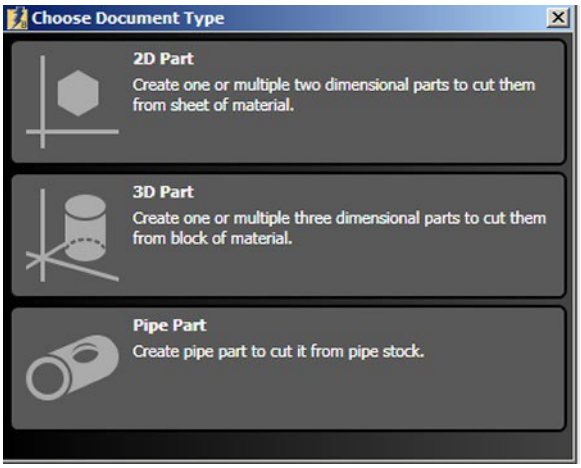
The **File** menu presents the following commands:

- New (Ctrl+N)
- Open (Ctrl+O)
- Close
- Save (Ctrl+S)
- Save as
- Save all
- Export
- Exit

FlashCut CAM adds two (2) File menu commands (see **FlashCut CAM** for details).

- Print Labels
- Print Nest

1. 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**.

2. Open (Ctrl+O)

Opens an existing CAD/CAM file for editing in a new FlashCut CAD window. It does not close the current drawing. FlashCut version 5.0 or later CAD/CAM files (file extension .cadcam) can be opened. However, Version 6.0 or later CAD/CAM files cannot be opened in an earlier version.

3. **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.

4. **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. These files are readable by FlashCut version 7 or later. The Save command is unavailable when no recent changes have been made.

5. **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. These files are readable by FlashCut version 7 or later.

6. **Save all**

   Saves all open drawings, including those in other windows. This command is unavailable when no recent changes have been made.

7. **Export**

   Export the file as a DXF/DWG file. There are two alternatives: Export DXF with Options or Export DXF.

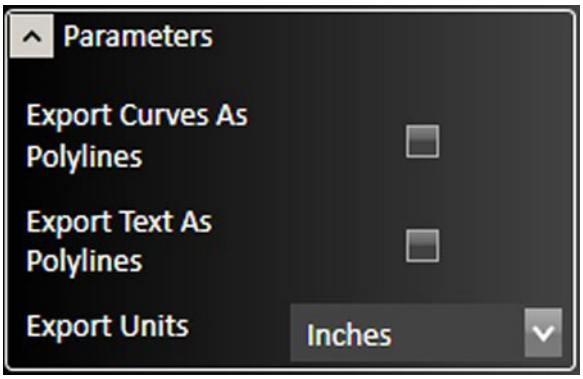
   Both commands bring up a Windows dialog box that allows you to select the location, filename, and specific file format.

   - 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.

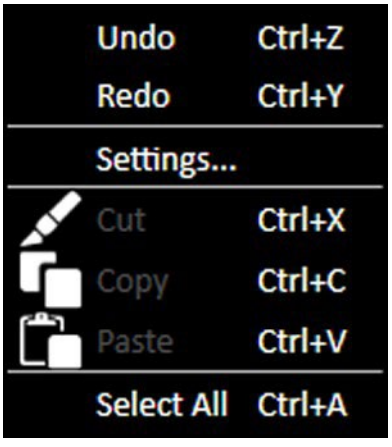
- Export Text as Polylines
  When selected, text shapes will be saved as polylines in the DXF file. Polylines are drawing objects composed of multiple separate line segments. When not selected, text will be saved as a font.
- Export Units
  Choose from either inches or millimeters.

8. Exit

Closes the entire FlashCut application. If any drawings are unsaved, FlashCut prompts you to save these files or discard changes before closing.

7.8.6 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)

1. **Undo (Ctrl+Z)**

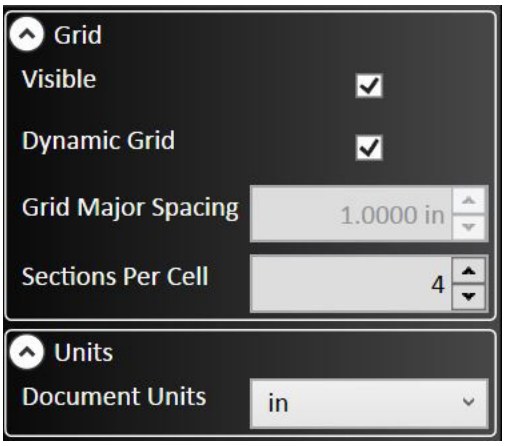
   Reverse the previous drawing action. Up to 20 actions can be reversed.

2. **Redo (Ctrl+Y)**

   Repeats the previous drawing action or reverses the Undo stack.

3. **Settings…**

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



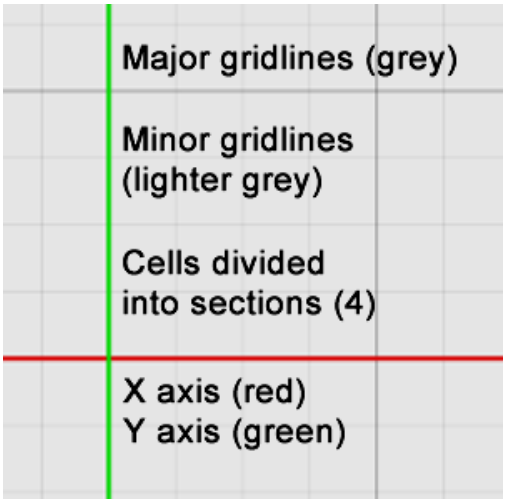
4. **Grid**

   - **Visible**

     The 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 dimensions that these gridlines represent remain the same while zooming and panning.



   - **Grid Major Spacing**

     Changes the size 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.

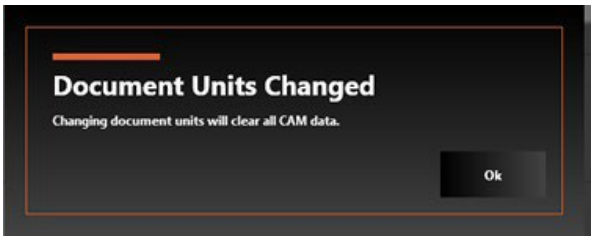
   - **Sections per Cell**

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

5. **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.



6. **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. You may also select all items in the workspace with the Select All (Ctrl+A) command.

7. **Copy (Ctrl+C)**

Copies the selected features and places them on the clipboard to be pasted.

8. **Paste (Ctrl+V)**

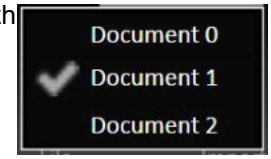
Pastes copied or cut features from the clipboard. The features will be centered at the cursor point and you will be able to maneuver them to a desired location. Clicking the mouse will paste the features permanently into the drawing.

9. **Select All (Ctrl+A)**

Use the Select All command to select all geometry in the workspace.

7.8.7 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.

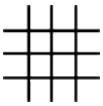
NOTE

You may copy features from one drawing and paste them into another drawing.

7.8.8 Part Shading and Grid Tools

These tools are displayed in the upper right corner of the drawing workspace in both FlashCut CAD and FlashCut CAM.

Toggles part shading in all open documents.



Toggles the grid in the active drawing workspace.

7.8.9 Parameters Area

Parameters for selected tools or objects are shown in this area in both FlashCut CAD and FlashCut CAM. Parameters displayed in this area will be described in the sections of this document where the tools or objects are discussed.

7.8.10 Status Bar

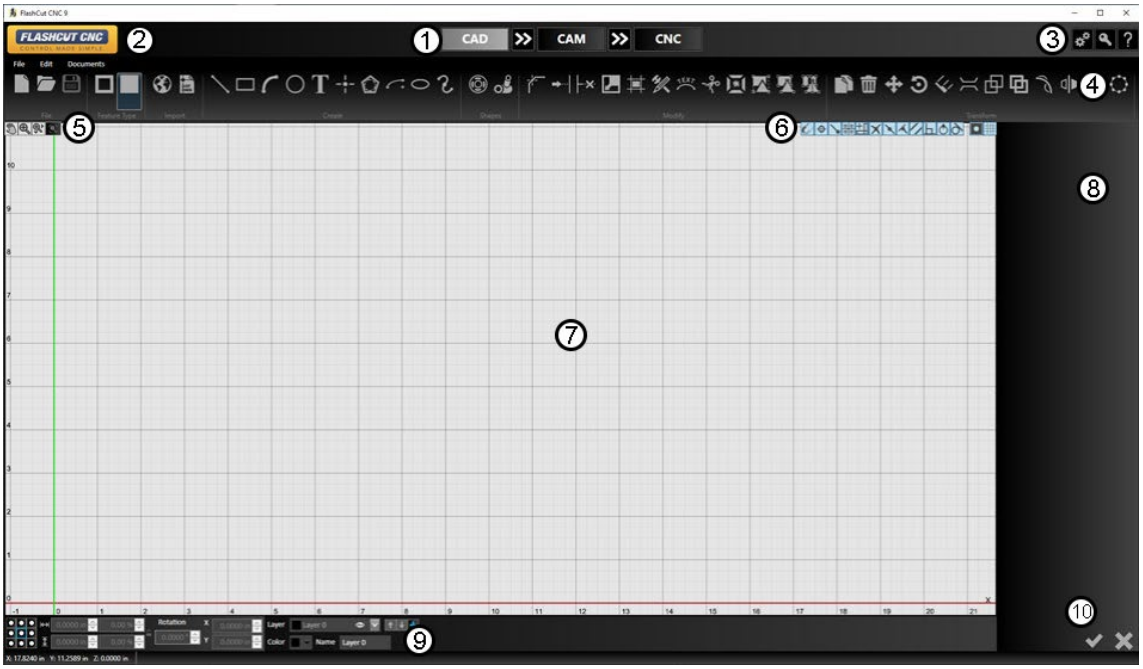
Reports the real time position of the cursor. In the CAD window, provides instructions to the user for the expected input of the currently selected tool.

NOTES:

8.0 FlashCut CAD

FlashCut CAD provides a fully-featured two-dimensional drawing environment for creating and editing drawings. This section of the manual provides a full description of all CAD commands and features.

The FlashCut CAD main screen is shown here.



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

1. Tabs
2. Menu bar
3. Configuration, License, and Help Buttons
4. Ribbon
5. Pan and zoom tools
6. Snap tools Part shading and Grid tools
7. Drawing workspace
8. Parameters area
9. Free form stretching and manipulation tools
10. Accept/Cancel

Several interface elements are the same in both CAD and CAM and are explained above. See Understanding common interface elements.

8.1 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.

There are six (6) different types of tools; each is explained below.

- File Tools
- Feature Type
- Create tools
- Shapes
- Transform tools

8.1.1 File Tools

There are three (3) file tools on the ribbon:

- New (Ctrl+N)
- Open (Ctrl+O)
- Save (Ctrl+S)

8.1.2 New (Ctrl+N)

Creates a new editable document in CAD. Available types are 2D part, 3D part, and Pipe Tube part.

8.1.3 Open (Ctrl+O)

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

8.1.4 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.

8.2 Feature Type

This pair of settings controls whether the top level of the drawing, indicated by the outermost line, is considered a part or a cut out. This setting will change which sections of the drawing are considered scrap, and which part is to be saved. The placement of kerf lines, lead ins, and corners is determined by this setting. For example, when making a circular cut in the workpiece, is the intention to make a disc, or to make a circular hole? The former is a part; the latter is a cutout. The behavior of nested closed shapes is also determined by this setting.

Changing from one to the other will reset the history of the Undo command.

8.2.1 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.

8.2.2 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.

This option is selected by default.

8.3 Import Tools

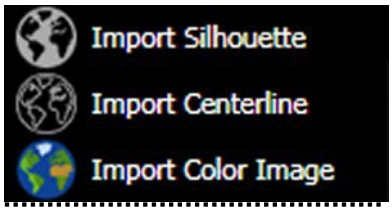
The import tools are used for adding images or DXF CAD drawings from an external file to the drawing. FlashCut accepts numerous image file types:

*.bmp	*.pgm
*.dib	*.ppm
*.jpg	*.sr
*.jpeg tif	*.ras
*.jpe	*.tiff
*.png	*.
*.pbm	

The are four (4) ways to import a drawing:

8.3.1 Import Silhouette, Centerline, or Color Image

Selecting the Import Tools icon from the ribbon brings up a menu with three (3) 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

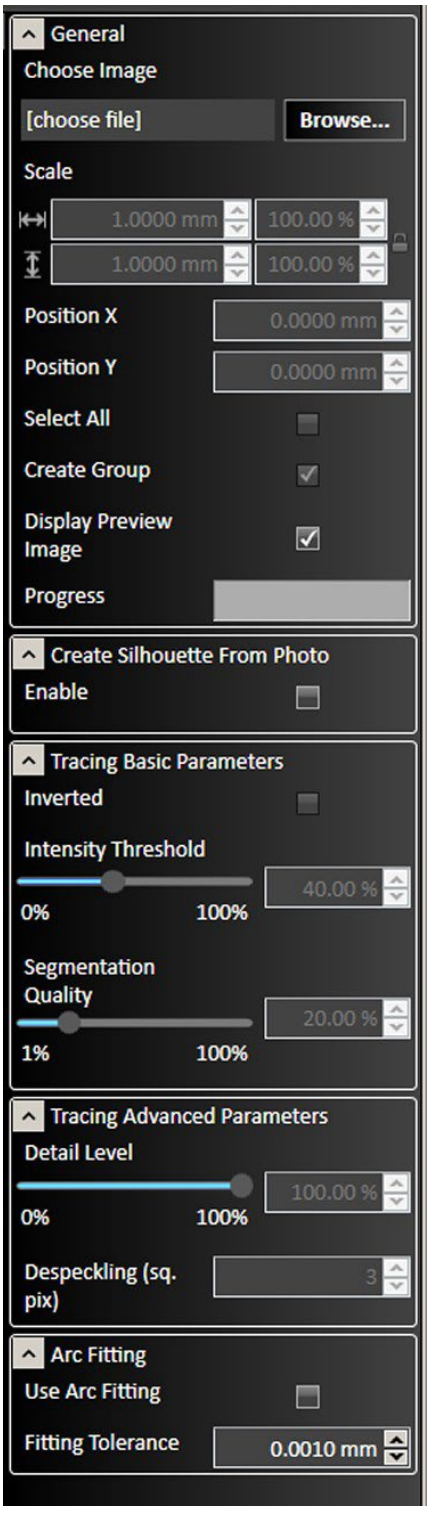
Import color image - creates an outline based on the colored areas of an image

8.3.2 Import DXF/DWG File

Import DXF/DWG image – imports lines and arcs from a DXF CAD drawing.

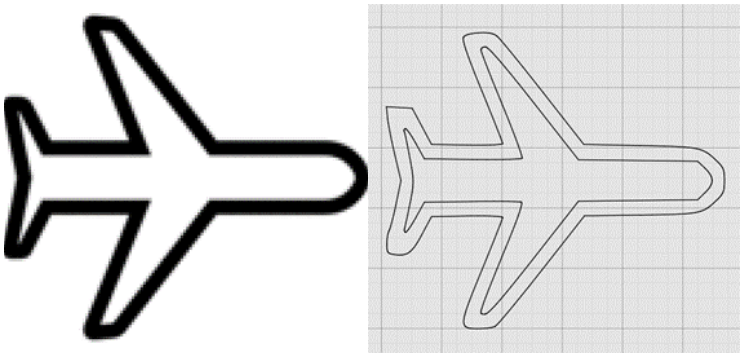
8.4 Understanding Silhouette and Centerline Images

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.



8.4.1 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.

1. **Choose Image**

   Click Browse… to select an image file from the computer. Select Open to call up the desired image. A list of file formats that can be imported appears on page 35.

2. **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.

3. **Position X**

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

4. **Position Y**

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

5. **Select All**

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

6. **Create Group**

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

7. **Display Preview Image**

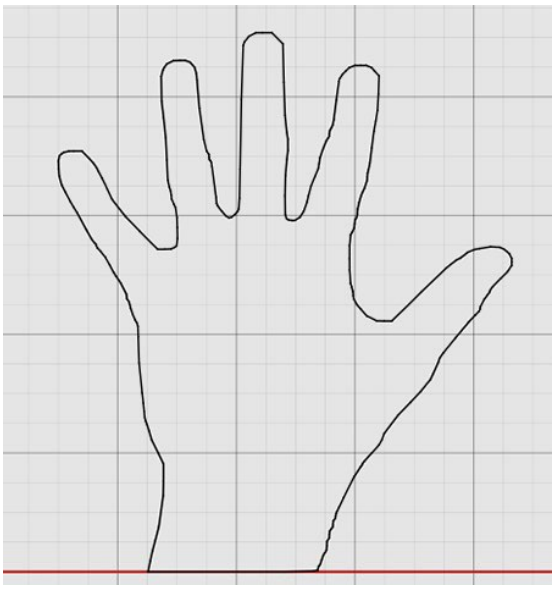
   Superimposes a translucent preview of the imported image while the new drawing is being generated.

8. **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 the rendering to finish between edits.

9. 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.

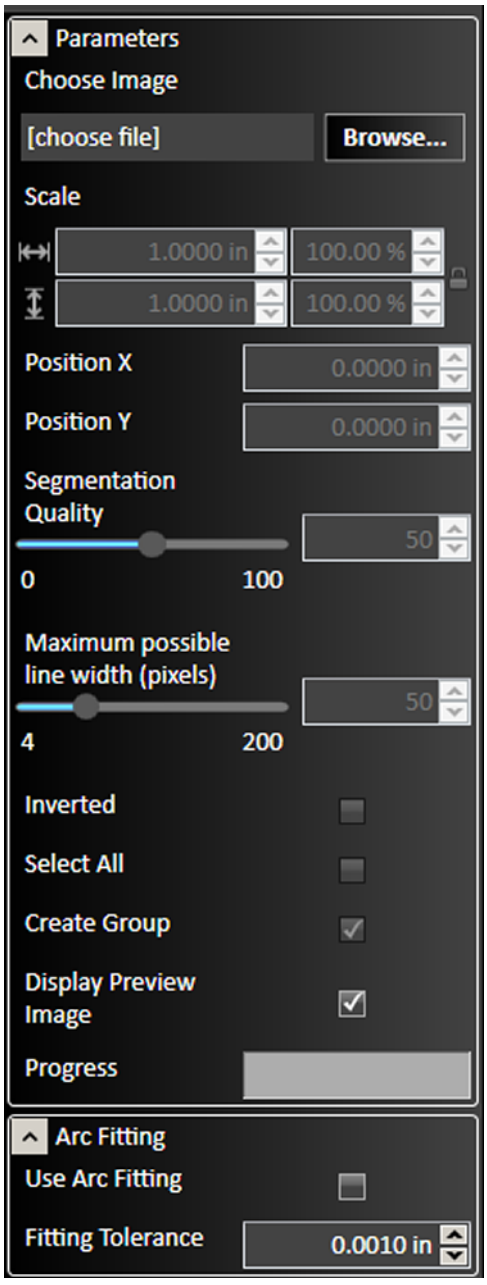
10. Tracing Basic Parameters

- **Inverted**
  When checked, this option reverses the shapes enclosed by the curves of the silhouette. For example, both objects below were imported as silhouettes from a photo.

The right-hand object is an inverted version of the left. The left-hand object is a part; the right-hand image is a cutout.

- **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.



11. 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.

12. Arc Fitting

- **Use Arc Fitting**

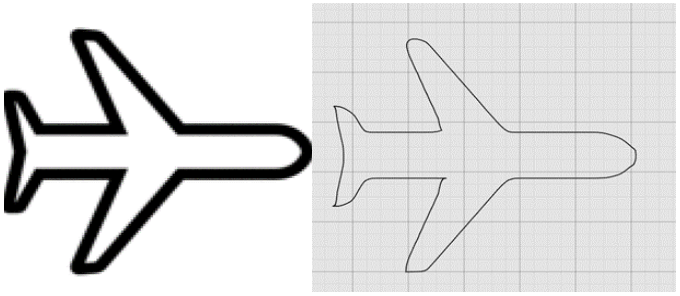
  When selected, **Arc Fitting Tool** will be applied to the imported image.

- **Fitting Tolerance**

  The **Tolerance** can be set in drawing units. A lower number will increase accuracy.

8.5 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.

1. **Choose Image**

   Click Browse… to select an image file from the computer. Click Open to call up the desired image. A list of file formats that can be imported appears on page 35.

2. **Scale**

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

3. **Position X**

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

4. **Position Y**

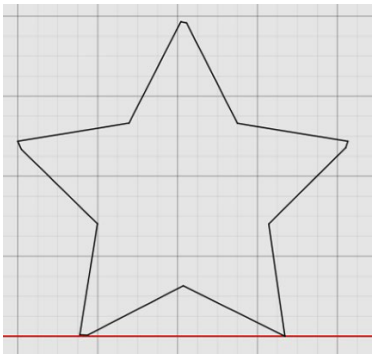
   Enter the value for the Y coordinate of the lower left corner of the imported image.

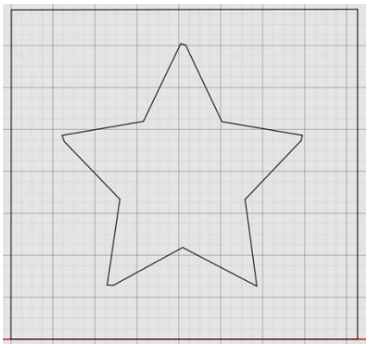
5. **Segmentation Quality**

   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.

6. **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).





7. **Select All**

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

8. **Create Group**

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

9. **Display Preview Image**

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

10. **Progress**

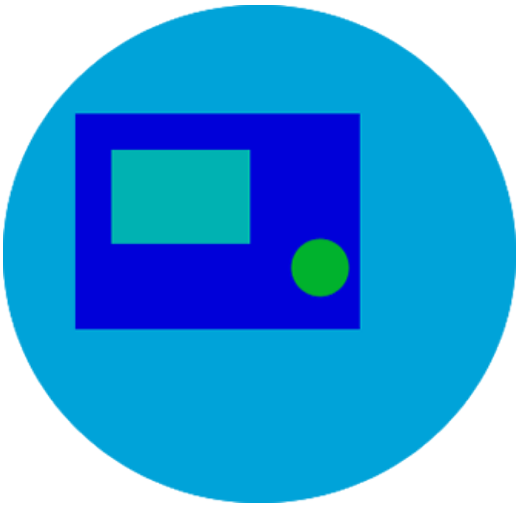
    Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for rendering to finish between edits.

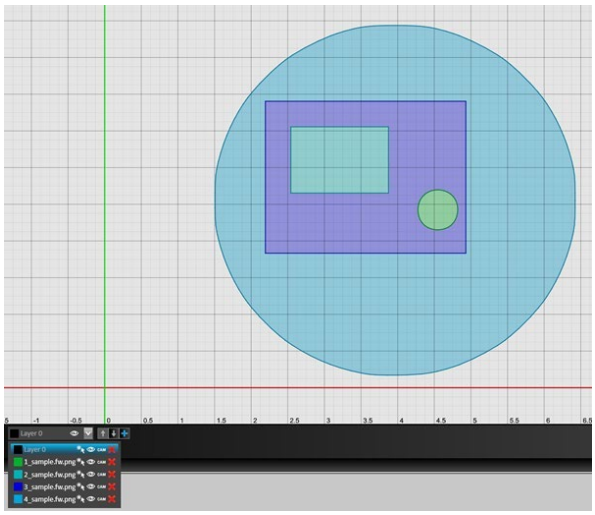
11. **Arc Fitting**

    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.

8.6 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 displaying and manipulating drawing objects using layers.





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

1. **Choose Image**

   Click Browse… to select an image file from the computer. Click Open to call up the desired image. A list of file formats that can be imported appears on page 35.

2. **Scale**

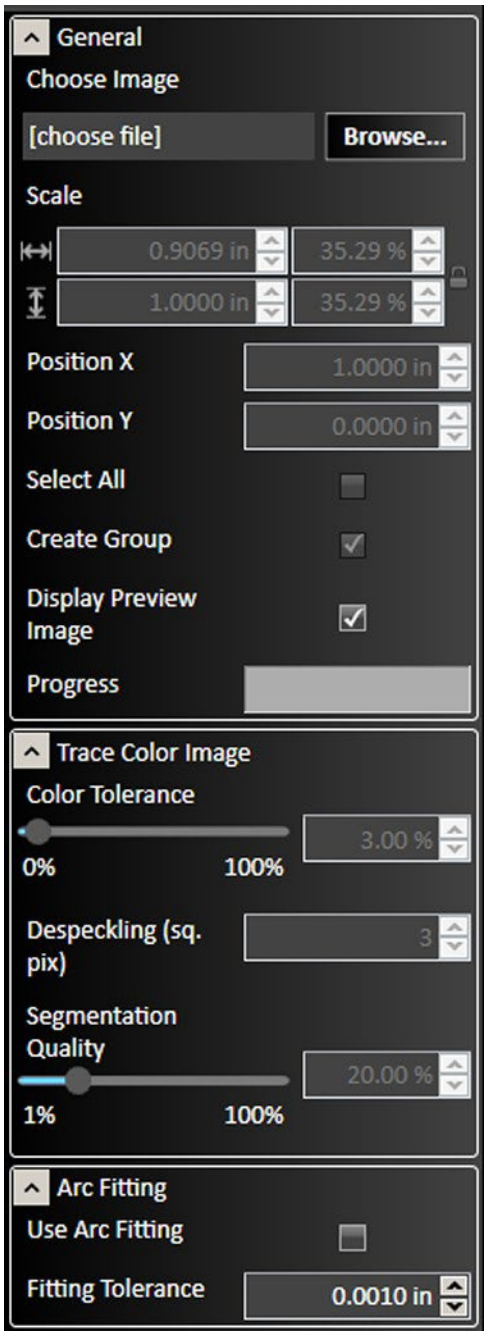
   Select how large or small the image appears in the drawing relative to its original size.

3. **Position X**

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

4. **Position Y**

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



5. **Select All**

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

6. **Create Group**

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

7. **Display Preview Image**

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

8. **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.

9. **Color Tolerance**

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

10. **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.

11. **Segmentation quality**

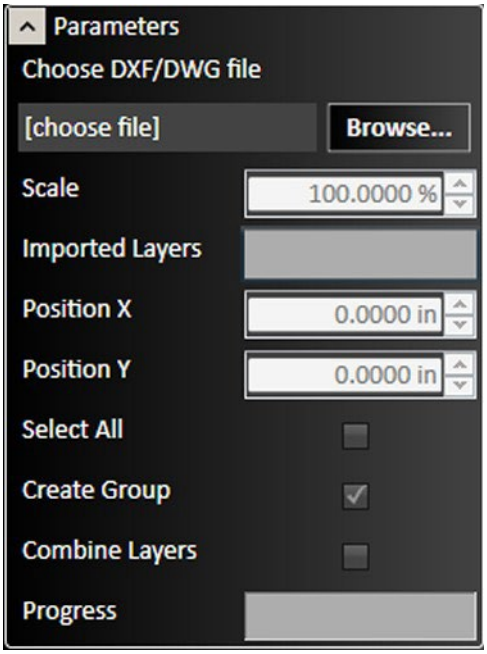
    Select how finely the program will divide curves. FlashCut CAD automatically breaks curves into separate line segments. When segmentation quality is increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing. Range 1-100.

12. **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.

8.7 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.

1. **Choose Image**

   Click **Browse**… to select a DXF file from the computer. Click **Open** to call up the desired image.

2. **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.

3. **Imported Layers**

   Select which layers from the DXF file you want to appear in the drawing.

4. **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.

5. **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.

6. **Select All**

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

7. **Create Group**

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

8. **Combine Layers**

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

9. Progress

Shows progress of file import.

8.8 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.

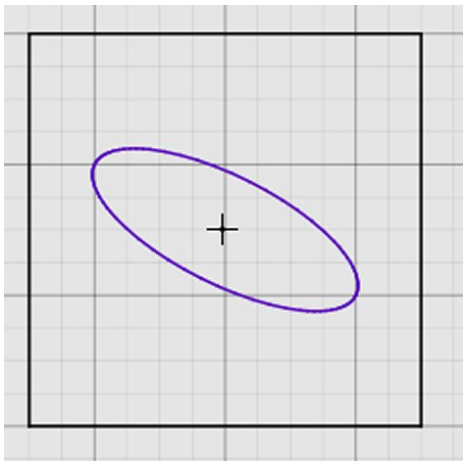
Each tool (with the exception of Point) lets you define the Treatment parameter to determine whether the element is cut or only marked.

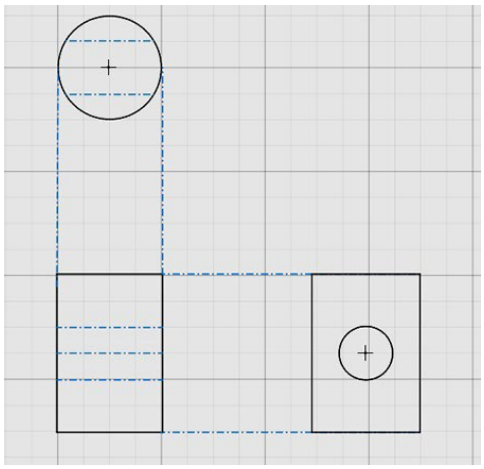
Select Cut Element to use the default cutting Fab Head. This is the pre-selected option. These elements appear in the CAD window as solid black lines.

Select Mark Element to use the default marking Fab Head. 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 (3) views.





Different creation modes exist for each tool, and each mode requires different information about the element being created. When creating an element graphically, or with the mouse, the information is required in a specific sequence. The **Status bar** provides step by step instructions on how to use the selected tool in a particular creation mode.

Note that the behavior of all of the Create tools is influenced by the active Snap tools. See the **Snap tools** section of the manual for details.

These are the tools that are available on the Create portion of the ribbon:

- Line
- Rectangle
- Arc
- Circle
- Text
- Point
- Polygon
- Elliptic Arc
- Ellipse
- Spline

8.8.1 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.

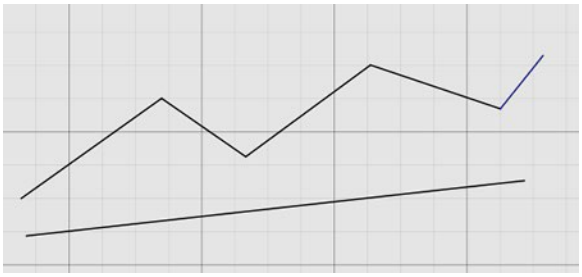
When creating a Two Point Line, the properties may be adjusted manually in the parameters window. Any single line segment (e.g., any segment that is part of a polygon or a multi-segment line) may be selected and its properties displayed.

1. **Creation Mode**

   Select the type of line to create.

   - Continuous Line
   - Two Point Line

   Examples of each type are shown below.



2. **Position, Offset, Angle and Length**

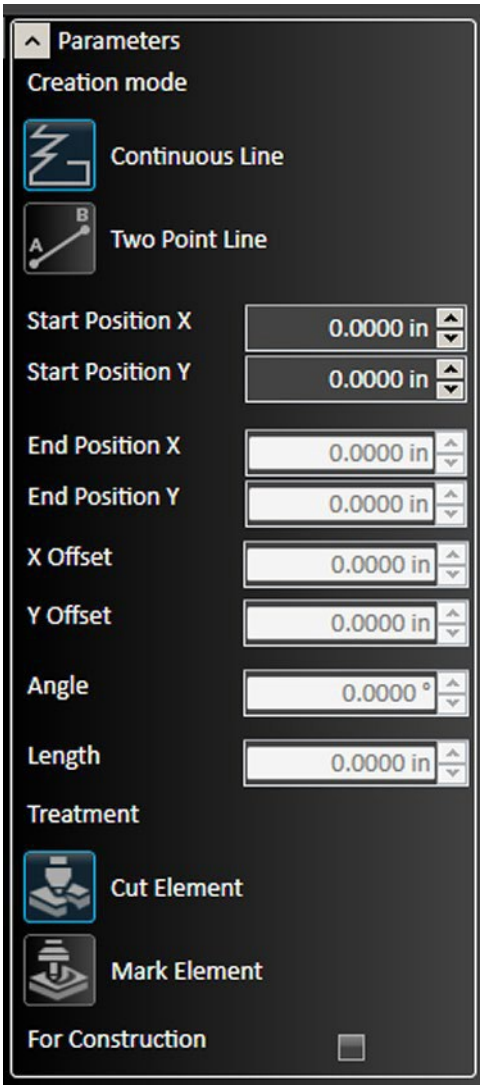
   Specify the attributes of a specific line segment.

3. **Treatment**

   Select whether the element should be cut or marked.

4. **For Construction**

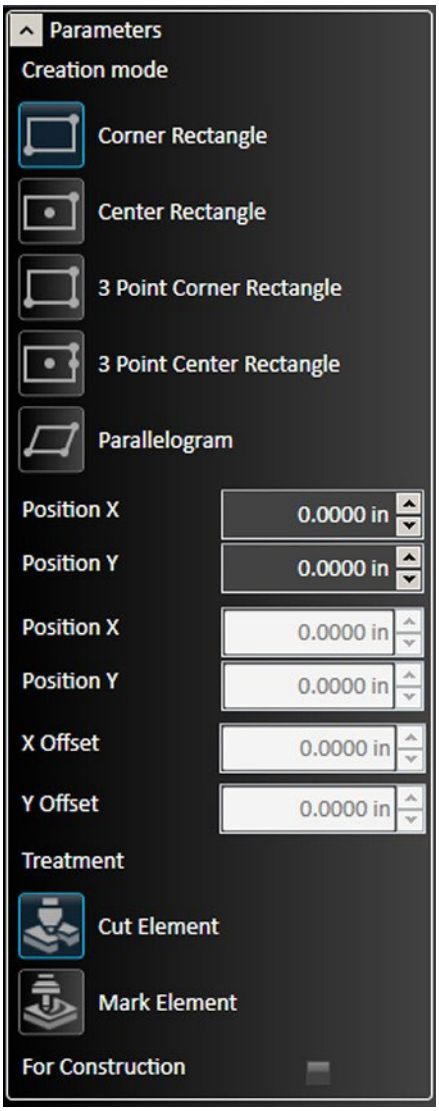
   Select to make the line a construction line.



8.8.2 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.



1. **Creation mode**

   Select the type of rectangle to create.

   - Corner Rectangle

   Drag from a single corner point.

   - Center Rectangle

   Drag from a center point.

   - 3 Point Corner Rectangle

   The first two points clicked describe one side. Drag to set the length of the remaining sides.

   - 3 Point Center Rectangle

   The first two points clicked set the length of the center line. Drag to set the distance from the center line to the lines parallel to it.

   - Parallelogram

   The first two points set the length of one side. Drag to set the position of the line parallel to the first.

2. **Position, offset, midpoint, corner**

   Specify the attributes of a specific point.

3. **Treatment**

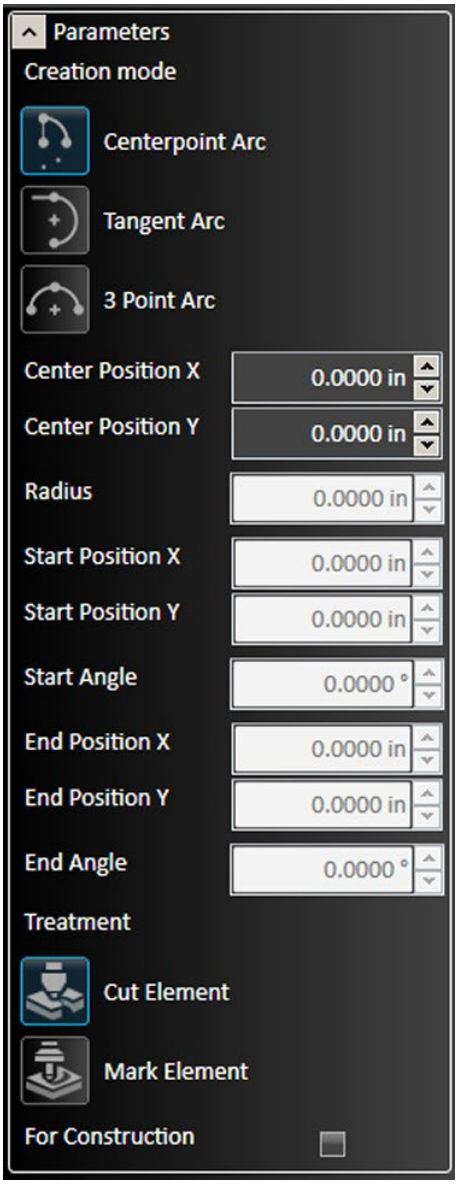
   Select whether the element should be cut or marked.

4. **For Construction**

   Select to make the rectangle a construction line.

8.8.3 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.

1. **Creation Mode**

   Select the type of arc to create.

   - **Centerpoint Arc**

   Click to define the centerpoint, drag to set the radius, drag to set the length.

   - **Tangent Arc**

   Click on the end point of an existing entity (point, line, rectangle, etc.). Drag to set the end point of the arc.

   - **3 Point Arc**

   Click and drag to set each point.

2. **Positions, Offsets, Angles, and/or Radius**

   Specify the attributes of the arc.

3. **Treatment**

   Select whether the element should be cut or marked.

4. **For Construction**

   Select to make the arc a construction line.

8.8.4 Circle

FlashCut CNC offers two (2) different options 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 (3) points on the circumference, through which the circle will be drawn.

1. **Creation Mode**

   Select type of line to create.

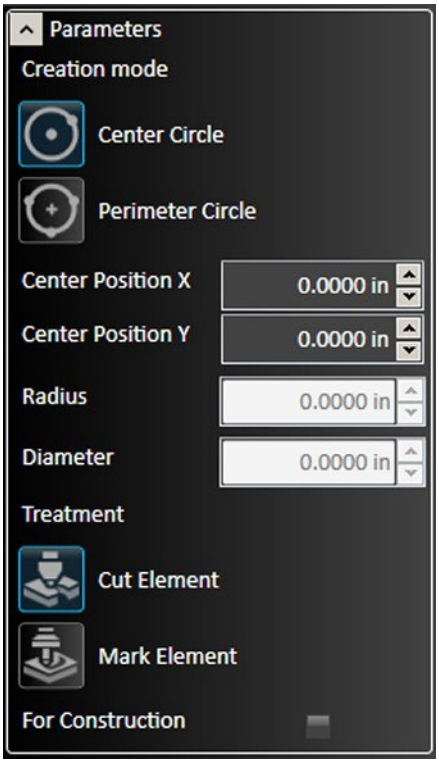
   - Center Circle

     Click to set the center point, drag to set the radius.

   - Perimeter Circle

     Click and drag to set three (3) points that define the perimeter of the circle.

   Examples of each type are shown below.



2. **Position, Radius, and Diameter**

   Specify the attributes of the circle.

3. **Treatment**

   Select whether the element should be cut or marked.

4. **For Construction**

   Select to make the line a construction line.

8.8.5 Text

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.

1. **Position X and Position Y**

   Specify the position of the anchor point relative to the origin.

2. **Text**

   Type the desired message in the text field. Can contain multiple lines.

3. **Font Name**

   Displays the name of the current font. Any installed TrueType font can be selected.

4. **Alignment**

   Select the alignment of the text relative to the anchor point. Options include left, right, and center.

5. **Size**

   Specify the size of the text object. Text objects are scalable. The size is shown in system units.

6. **Kerning Offset**

   Specify the spacing between letters.

7. **Line Spacing**

   Select a value between 50% and 300%.

8. **Angle**

   Specify the angle relative to the anchor point. 360-degree rotation in either direction permitted.

9. **Bold and Italic**

   Check box to apply style.

10. **Treatment**

    Select whether the element should be cut or marked.

11. **For Construction**

    Select to make the text object a construction line.

8.8.6 Point

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

8.8.7 Polygon

FlashCut CNC enables you to select the number of sides when creating a polygon.

1. **Center X/Y Position**

   Specify the center of the polygon relative to the origin.

2. **Sides Count**

   Defines the type of polygon. Minimum number of sides is three.

3. **Internal Diameter**

   Diameter of an interior circle tangent to all sides (i.e., the construction circle).

4. **External Diameter**

   Diameter of an exterior circle tangent to all vertices.

5. **Side Length**

   Length of each side.

6. **Leave Construction Circle**

   Causes the circle to remain in the drawing after the polygon has been completed.

7. **Treatment**

   Select whether the element should be cut or marked.

8. **For Construction Option**

   Select to make the polygon a construction line.

8.8.8 Elliptic Arc

Creates an arc that is a section of an ellipse.

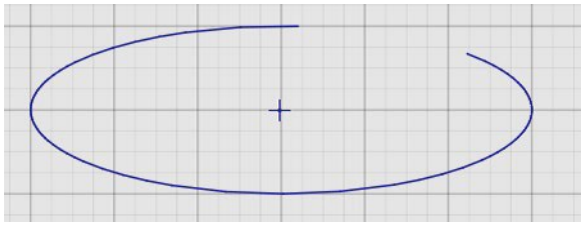
1. **Drawing the Arc**

   Click in the drawing window to select a center point and the first radius of the ellipse that will remain fixed.

Move and click the cursor to determine the second radius of the ellipse.

2. **Center Position X and Y**

   Specify the coordinates of each center point.



3. **Radius 1 and Radius 2**

   In the example below, Radius 1 is the radius of a circle defined by the center point and the X-Axis, while Radius 2 is the radius of a circle defined by the center point and the Y-Axis.

4. **Major Axis Angle**

   Changing this value allows the arc to be rotated around its center point.

5. **Start and End Angle**

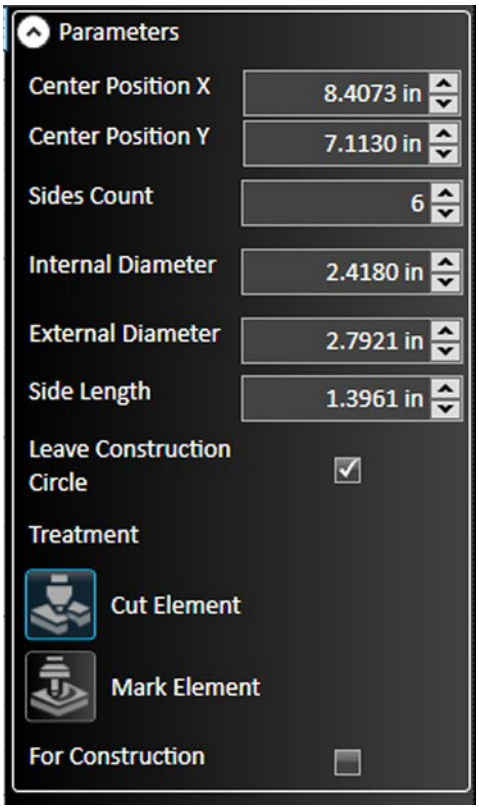
   Select whether the element should be cut or marked.

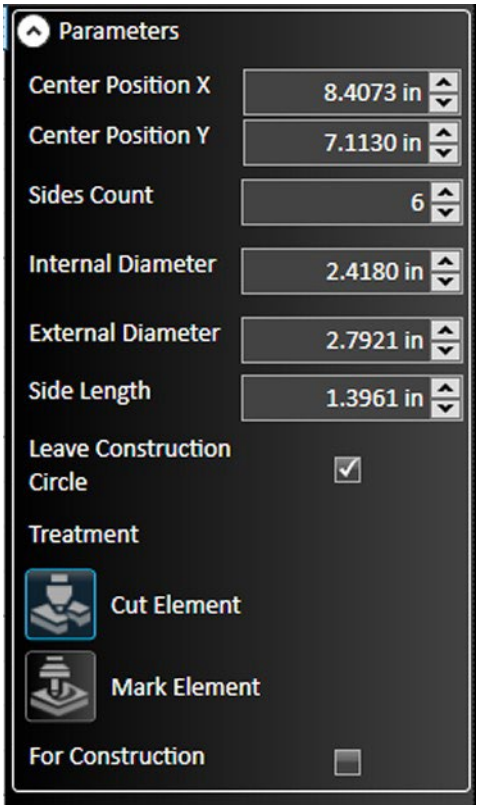
6. **Treatment**

   Select whether the element should be cut or marked.

7. **For Construction Option**

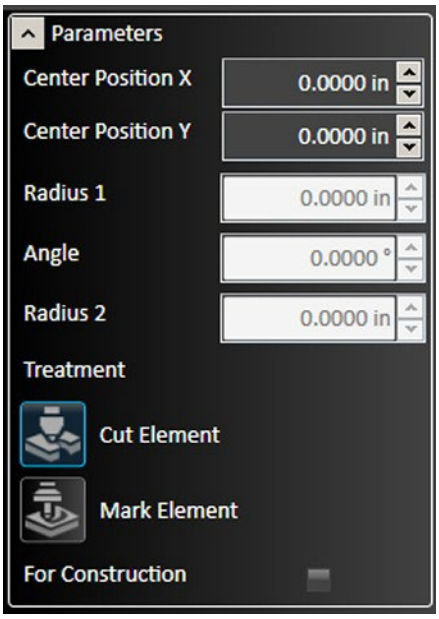
   Select to make the line a construction line.





8.8.9 Ellipse

Creates an ellipse.



1. **Drawing the Ellipse**

   Click in the drawing window to select a center point and the first radius of the ellipse that will remain fixed.

   Move the cursor to determine the second radius of the ellipse and click for the desired bend.

2. **Center Position X and Center Position Y**

   Specify the coordinates of each center point.

3. **Radius 1 and Radius 2**

   Values defined by the center point and the furthest extent of the ellipse along each axis.

4. **Angle**

   Changing this value allows the ellipse to be rotated around its center point.

5. **Treatment**

   Select whether the element should be cut or marked.

6. **For Construction**

   Select to make the ellipse a construction line.

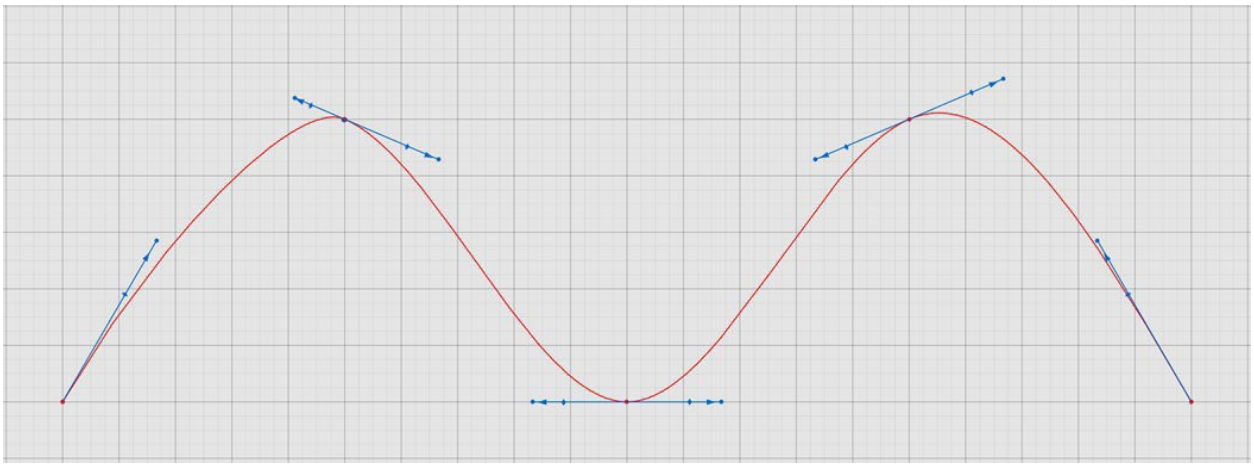
8.8.10 Spline

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.

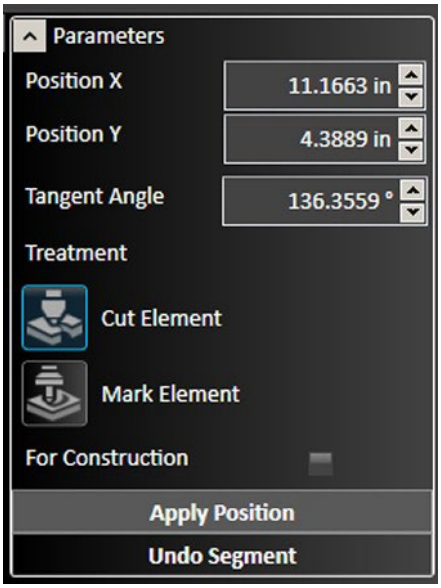
1. **Drawing each curve segment**

   Click the start point of the curve and then select subsequent points through which the spline will pass.



2. **Position X and Position Y**

   Specify the attributes of a specific line segment.



3. **Treatment**

   Select whether the element should be cut or marked.

4. **For Construction**

   Select to make the line a construction line.

5. **Apply Position**

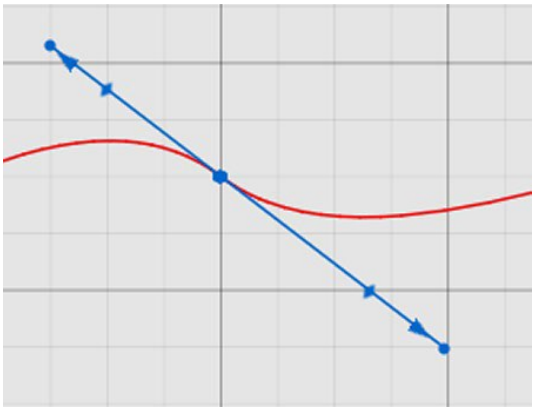
   The default function of each click: the new line segment is added.

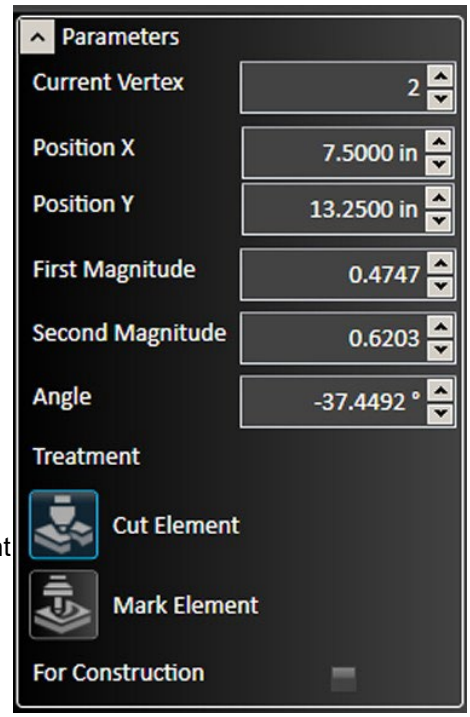
6. **Undo Segment**

   When clicked, removes the segment just added. Consecutive clicks remove additional existing segments.

7. **Adjusting a curve segment**

   The example below shows a single vertex. Rotate or extend the blue control line to change the shape of the curve at the vertex.





8. **Current Vertex**

   Select the particular vertex on the spline.

9. **Position X and Position Y**

   Specify the coordinates of the vertex.

10. **First Magnitude and Second Magnitude**

    Shows the length of each control arrow. In the example above, the arrow representing the first magnitude points to the upper left.

11. **Angle**

    Describes the angle of the blue control line tangent to the curve at the vertex.

12. **Treatment**

    Select whether the element should be cut or marked.

13. **For Construction**

    Select to make the line a construction line.

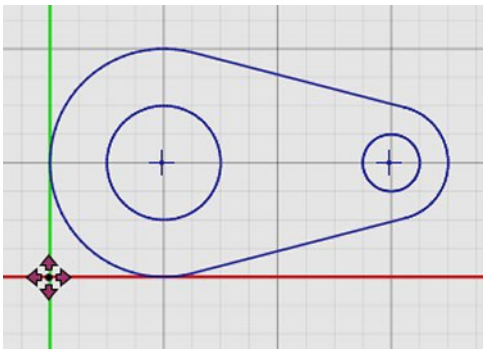
8.9 Shapes

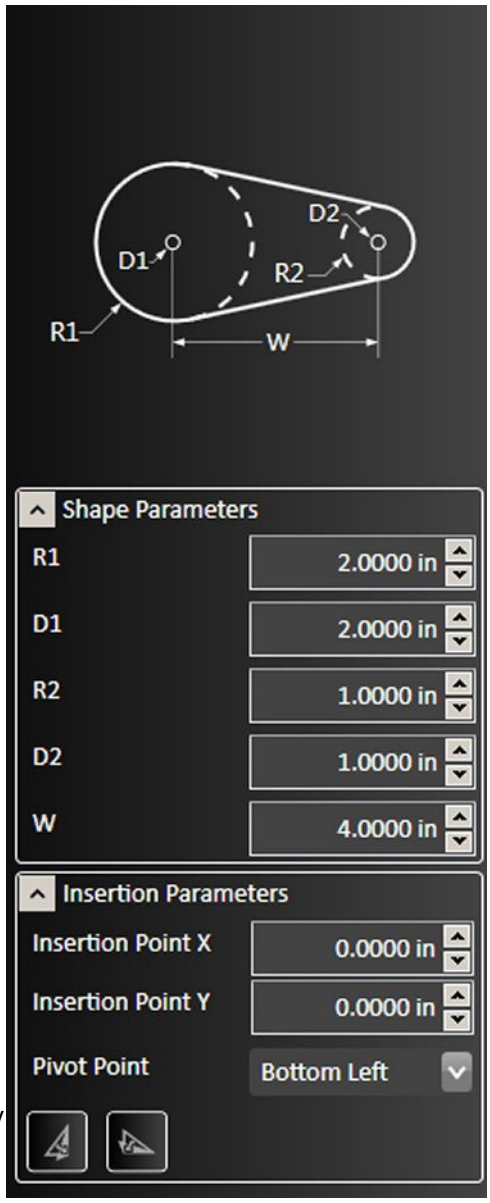
8.9.1 Shape Tool

FlashCut CNC offers a library of shapes for both simple and complex parts. This library is part of the pro package and is also offered as an a la carte feature. Select the shape tool from the ribbon to load.

To choose a specific shape, select it from the list.

8.9.2 Example: Two (2) circles connected by tangents





1. **Shape Parameters**

   When the shape is imported into the drawing window, its specific parameters appear. For example, for the shape above any of the following attributes can be manually adjusted: R1 and R2 (the radii of the two circles), D1 and D2 (the diameters of the two through holes), or W (the distance between the center point of the two circles).

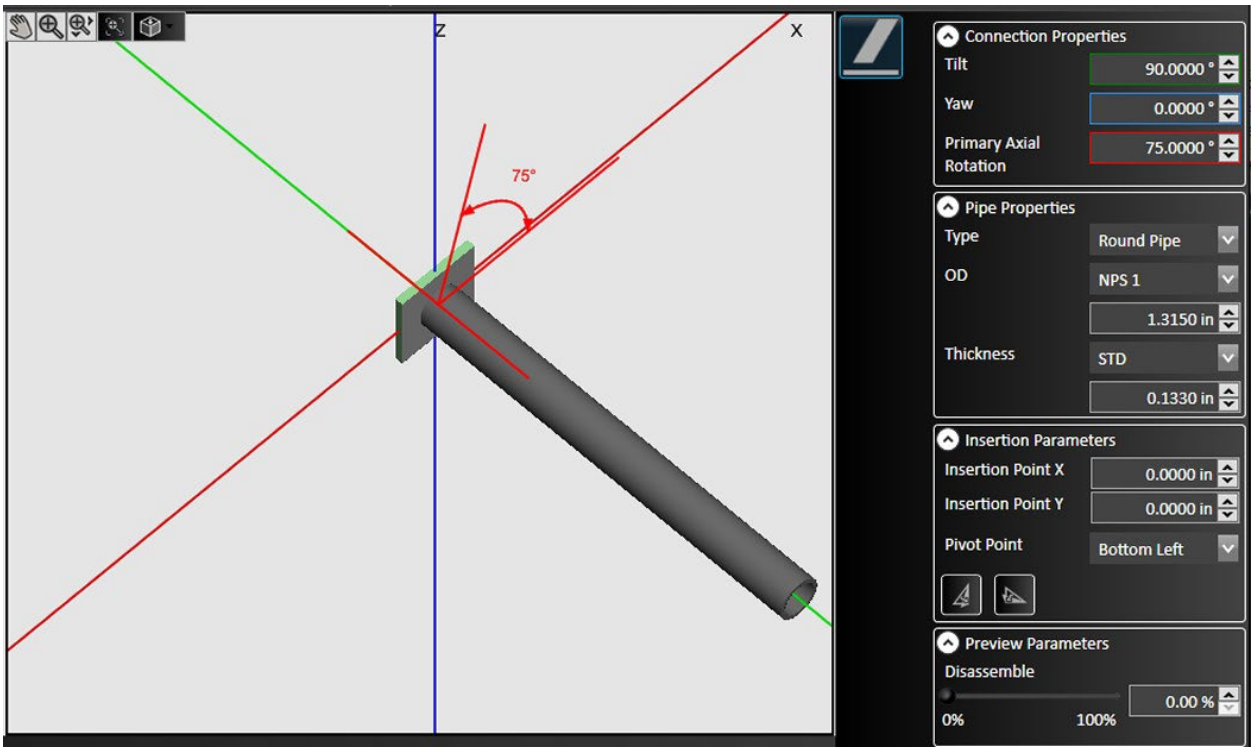
2. **Insertion Parameters**

   The position of the shape on the page and the location of its pivot point can be selected. In addition, the shape can also be flipped horizontally or vertically.

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

8.9.3 Pipe Joint Shape Tool

FlashCut CNC offers a set of preconfigured alternatives for visualizing and manipulating pipe joints, in drawings that contain pipe parts. The figure below shows the 3D preview window and the parameters pane.



8.9.4 Connection Properties

Configurable parameters include Tilt, Yaw, and Primary Axial Rotation in degrees.

8.9.5 Pipe Properties

Standard pipe dimensions are included. Configurable parameters include Pipe Type, OD (outer diameter), and Thickness, in drawing units.

8.9.6 Insertion Parameters

The position of the X/Y Insertion Points and the location of its Pivot Point can be specified. In addition, the shape can also be flipped horizontally or vertically.

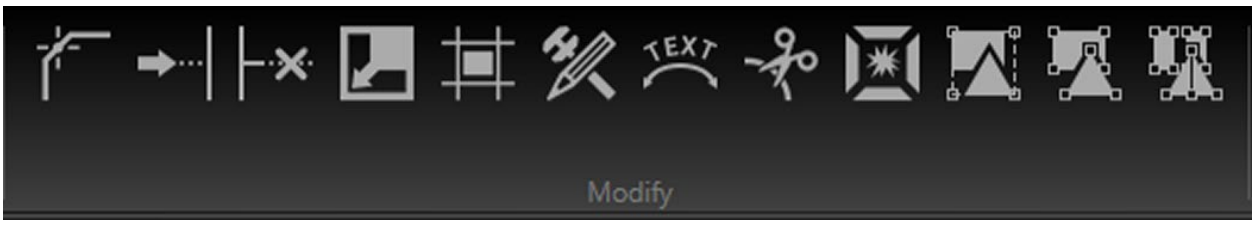
8.9.7 Preview Parameters

The components can be disassembled (offset in the preview) for easier visualization. The range is 0-100%.

8.10 Modify Tools

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

After modifications are complete, click the green check mark to Accept changes or the red X mark to Cancel changes.



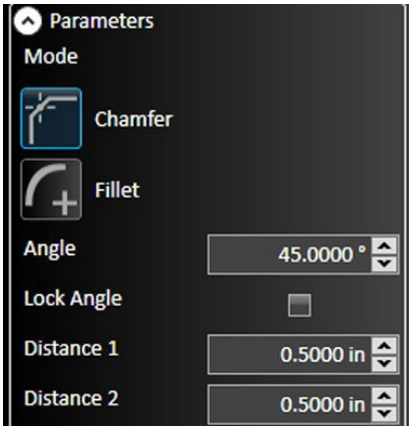
These are the tools that are available on the Modify portion of the ribbon:

Corner Tool	Apply Offset	Group
Extend Tool	Fix Drawing	Ungroup
Trim Tool	Curve Text	Ungroup All
Scale Entitles	Cut Tool	
	Explode Entities	

8.11 Corner Tool

The corner tool allows you to apply chamfers or fillets to shapes.

8.11.1 Apply Chamfer

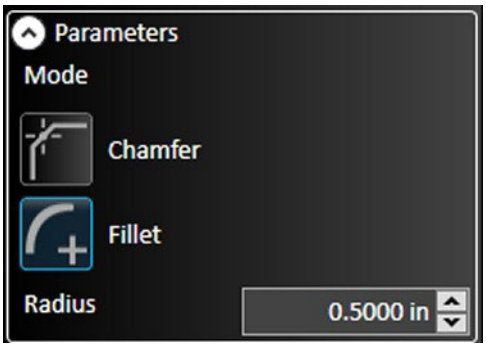


Select two (2) intersecting lines or their intersection point and then click Chamfer to add a sloped chamfer to this edge. You can scale the size of the chamfer by dragging the mouse towards or away from the chamfer, or in the parameter window, type the distances from the intersection in both directions and the slop angle.

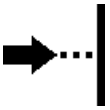
Choosing the Lock Angle option causes both distances to change when one of them id edited, holding the angle measurement constant. Once the chamfer is correctly configured choose the green check mark or press Enter.

8.11.2 Apply Fillet

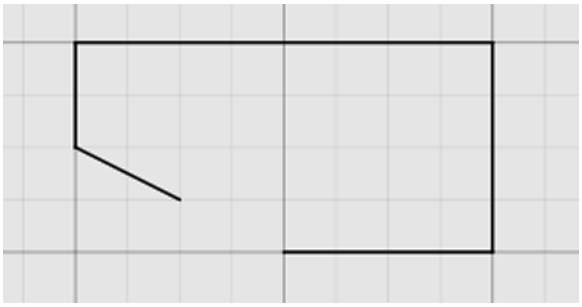
Select two intersecting lines or their intersection point and then click Fillet to add a rounded fillet to this edge. You can scale the size of the fillet by dragging the mouse towards or away from the fillet, or in the parameter window, type in the radius of the fillet. Once the fillet is correctly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

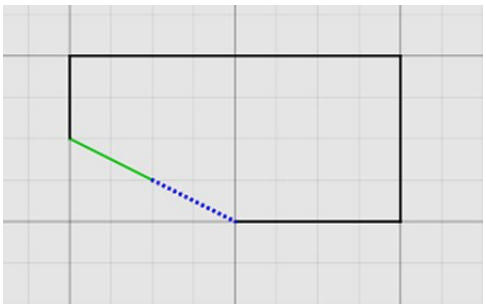


8.12 Extend Tool

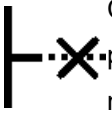


Click the Extend Tool and then hover the selection cursor over the endpoint of the line or curve to be extended. A preview of the extension should appear (in blue, below) to show where the line or curve will intersect with another in the drawing. Click the mouse to finalize the extension.

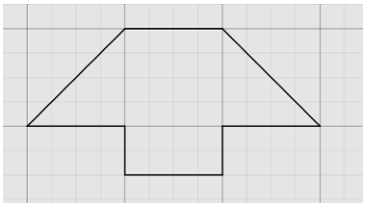


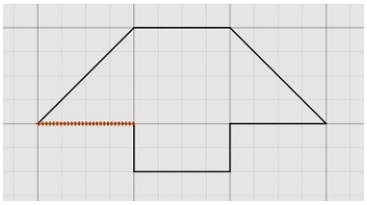


8.13 Trim Tool



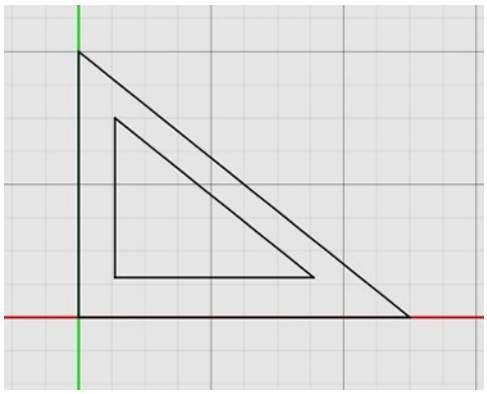
Click the Trim Tool and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear (in red, below) where the line or curve will be removed. Click the mouse to finalize the trim.

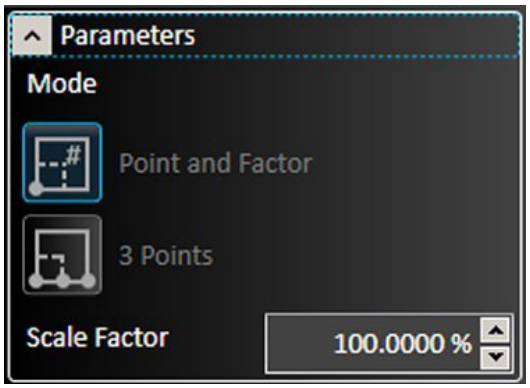


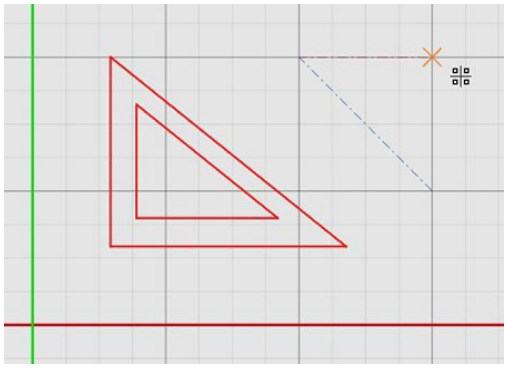


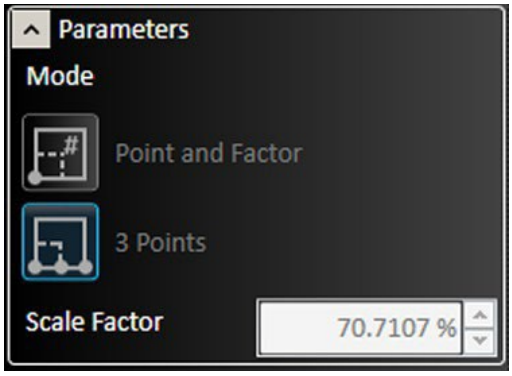
8.14 Scale Entities

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.



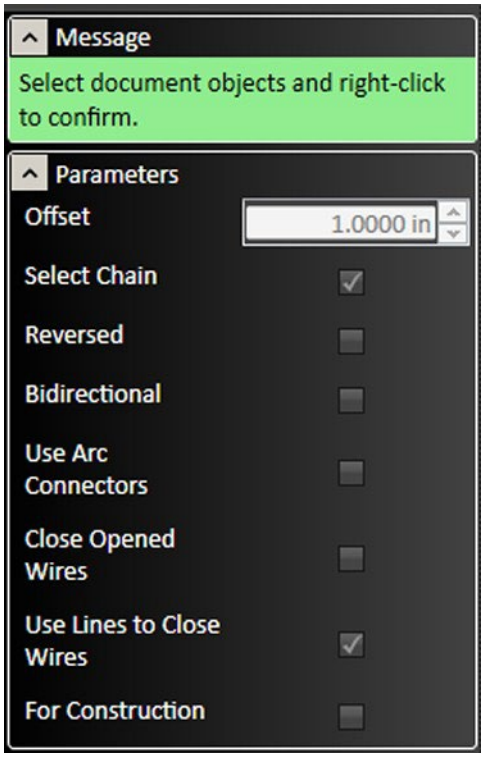






8.15 Apply Offset

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 position them at the specified distance away from the original feature.

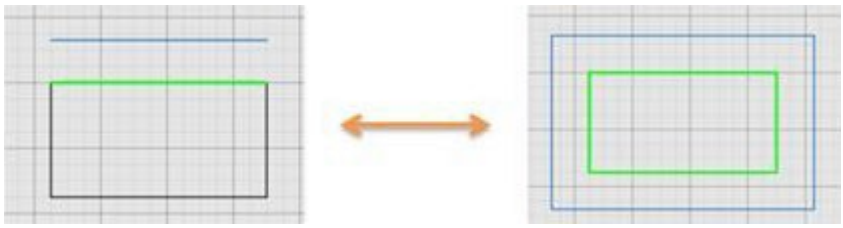
1. **Offset**

   The Offset distance indicates the distance from the original feature that the copy will appear.

2. **Select Chain**

   Select Chain causes the offset tool to select all touching segments of the feature originally chosen and create an offset to the resulting figure, rather than a single line. For instance, selecting one side of a box with Select Chain chosen creates an offset copy of the entire box as shown here.

   Note that you can also select a chain anywhere in the program by pressing the "alt" key while you click any segment in a chain.



Select Chain off **Select Chain on**

3. **Reversed**

   Reversed changes the direction of the offset distance.

4. **Bidirectional**

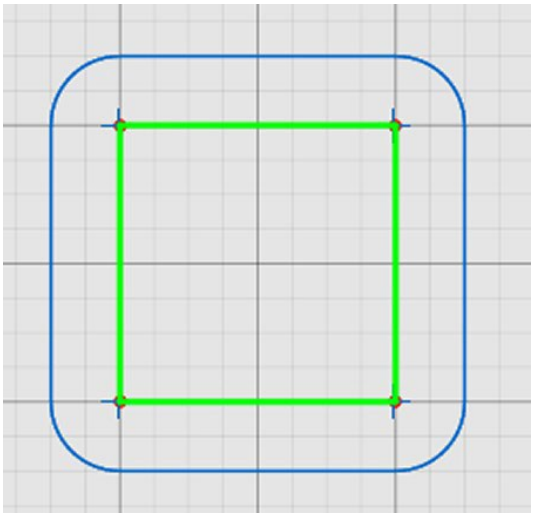
   Bidirectional creates a second copy, opposite the first. When the Bidirectional option is chosen, the Close Opened Wires option appears.

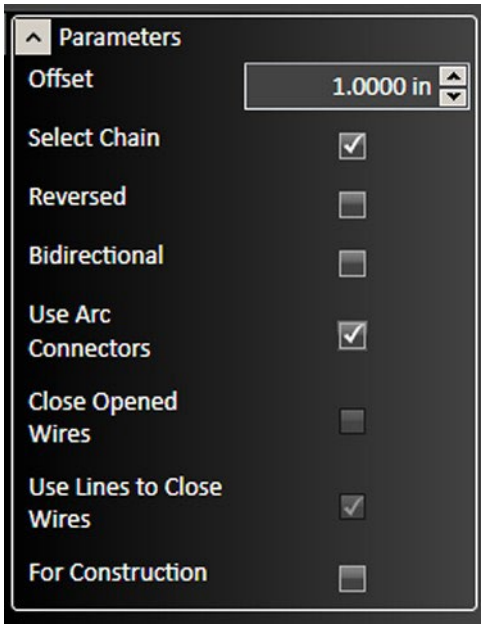
5. **Close Opened Wires**

   Close Opened Wires option closes the gaps created by the offset ends when chosen.

6. **Use Arc Connectors**

   Use Arc Connectors rounds out the corners of the offset by transforming them into arcs. An example appears below.





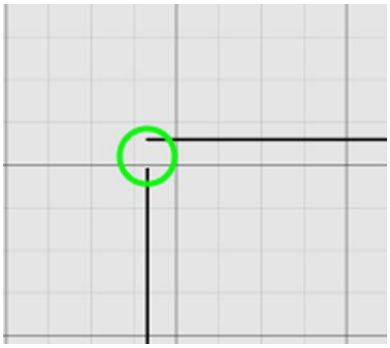
7. **Use Lines to Close Wires**

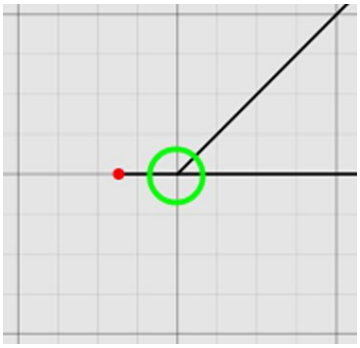
   Use Lines to Close Wires changes the rounded edges to flat, straight lines.

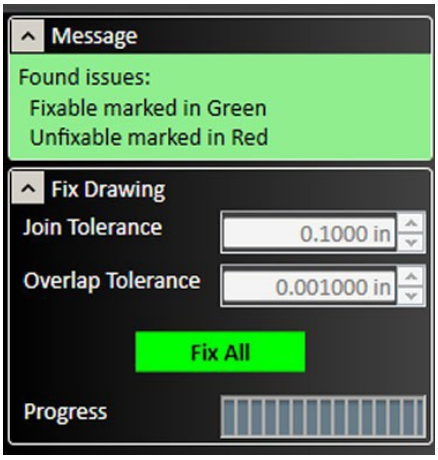
8.16 Fix Drawing

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. Two fixable errors appear below.

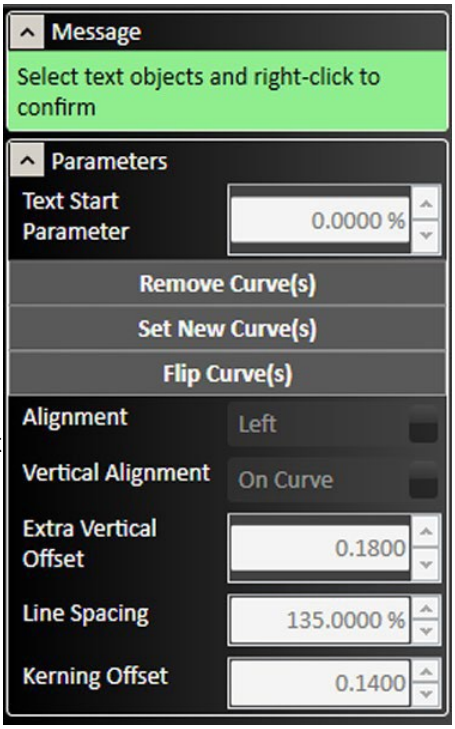






Use the Search Issues button to identify issues with the drawing that could lead to mistakes in fabrication. Issues noted with a green circle can be fixed automatically by selecting Fix All. Issues noted with a red circle must be addressed manually.

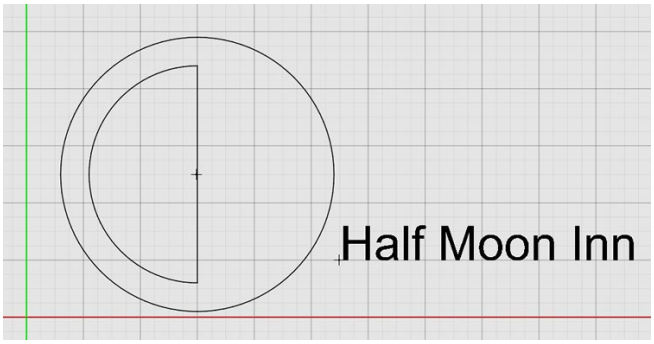
8.17 Curve Text

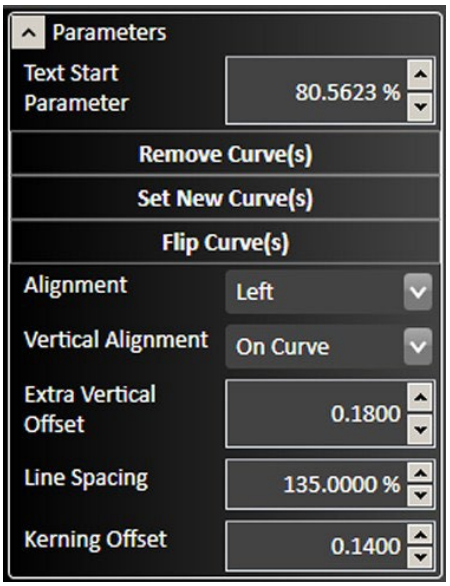


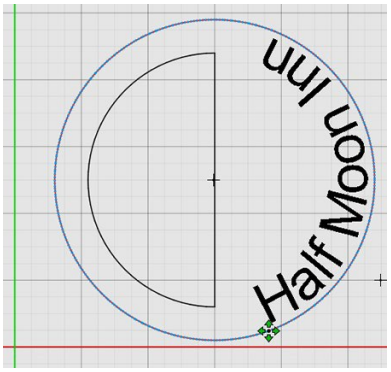
The Curve Text tool modifies an existing text object to follow a curved shape. Start by creating both the text object and the curved shape you want to use.

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.

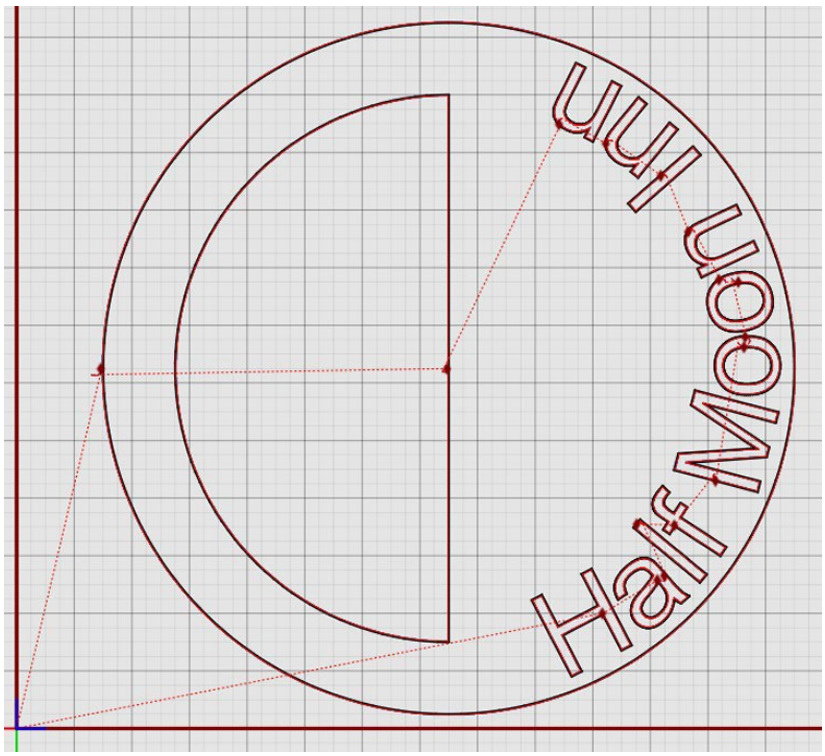
In the example below, note that the Extra Vertical Offset and Kerning Offset have both been adjusted to make the curved text even and legible.







The resulting CAM toolpath for the sample drawing appears below. Note that it is not necessary to explode letter shapes to create toolpaths.



- Text Start Parameter

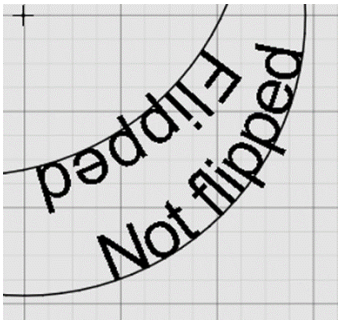
Indicates the position on the curve where the text begins. This can be set by adjusting the value in the parameters window, or by dragging the control handle in the drawing window.

- Remove Curve(s)

Separates the text object from the curve.

- Set New Curve(s)

Select a different curved object to combine with the text object.



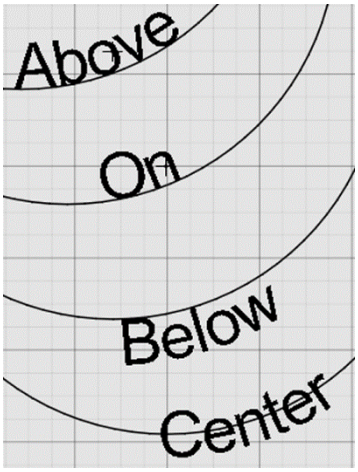
- Flip Curve(s)

Changes the side of the curve that the text object follows.

- Alignment

Describes the relationship between the start of the line of text and the control point. Options include:

- Left
- Right
- Center
- Vertical Alignment
    - Above Curve – Text always is above the line.
    - On Curve – Text follows the curve, but portions of the font may dip below the line.
    - Below Curve – Text is always below the curve.
    - Curve Center – The line passes through the center of the text.



- Extra Vertical Offset

Controls the relationship between the text objects and the curve.

- Line Spacing

Affects the spacing of multiple lines of text.

- Kerning Offset

Adjusts the spacing between letters.

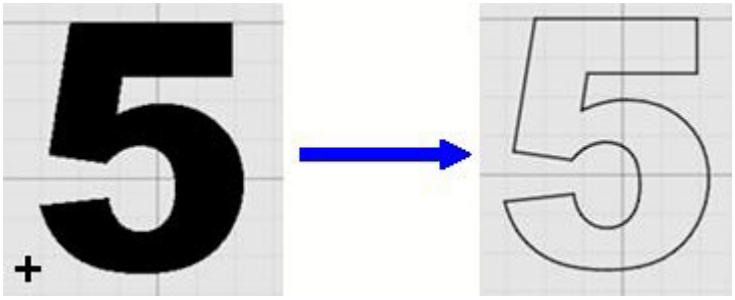
8.18 Cut Tool

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 of the cut worked properly.

8.19 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.

Select a solid feature and then click Explode Entities. See the example below.



8.20 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.

8.21 Ungroup

The Ungroup command separates the selected grouped objects.

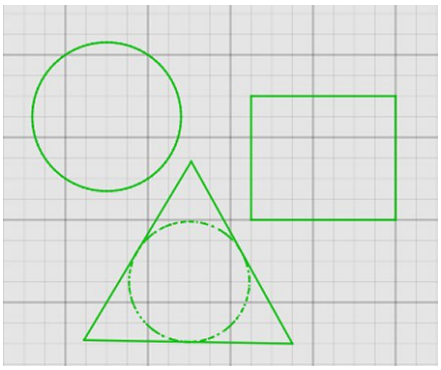
If you choose this command without a group selected you will be prompted to select the objects to be grouped and to confirm the action by right-clicking.

8.21.1 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.

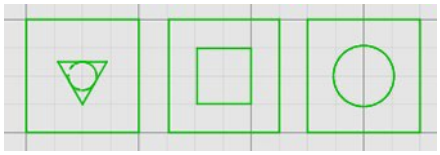
The example below shows the highlighting and message for all grouping operations.



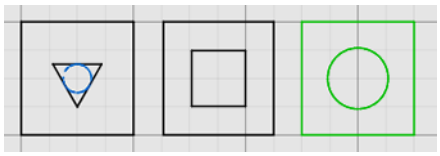
8.21.2 Ungroup vs. Ungroup All

The difference in behavior between Ungroup and Ungroup All is shown below. Note how the selections (green) indicate the level of grouping all, some, none.

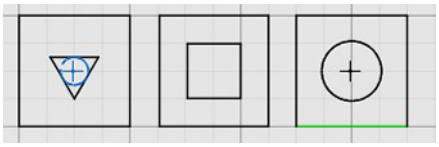
Each object in the group is itself a group of multiple objects.



- Ungroup (center) will separate the group but leave each subgroup intact.



- Ungroup All (bottom) will break each object into its component lines.



8.22 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. After transformations are complete, click the green check mark to Accept changes or the red X mark to Cancel changes.

These are the tools that are available on the Transform portion of the ribbon:

Copy Entities	Perform Boolean Operation on Entities
Delete Selected Objects	Weld Selected Entities
Move Entities	Arc Fitting Tool
Rotate Entities	Minor Entities
Measure Entities with Ruler	Linear Pattern Tool
Bridge Entities	Circular Pattern Tool

8.22.1 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.

8.22.2 Delete Selected Objects

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

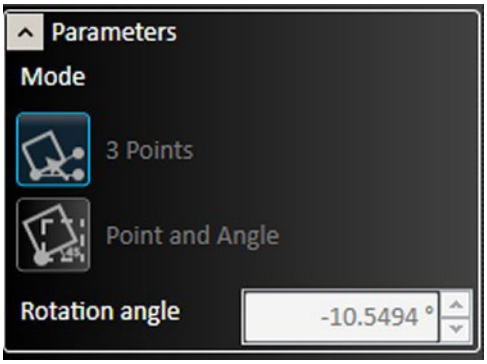
8.22.3 Move Entities

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 features (0) in relation to that point. Click the mouse again to select the new location for the features.

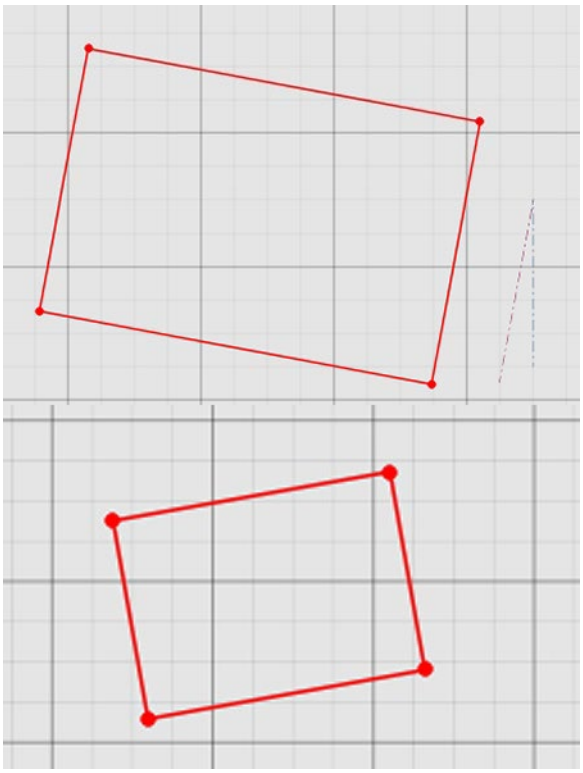
8.22.4 Rotate Entities

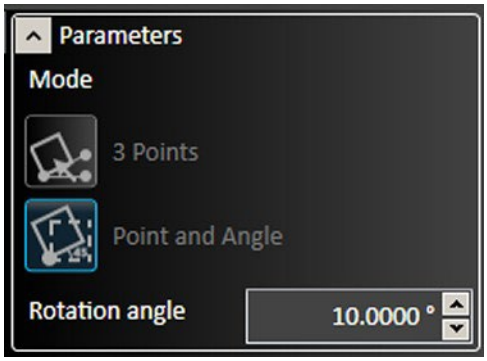
FlashCut offers two (2) modes for rotating features in relation to a set point.

The 3 Point rotation prompts you to select a center point, a point near the feature, and a final point to which the feature will rotate.



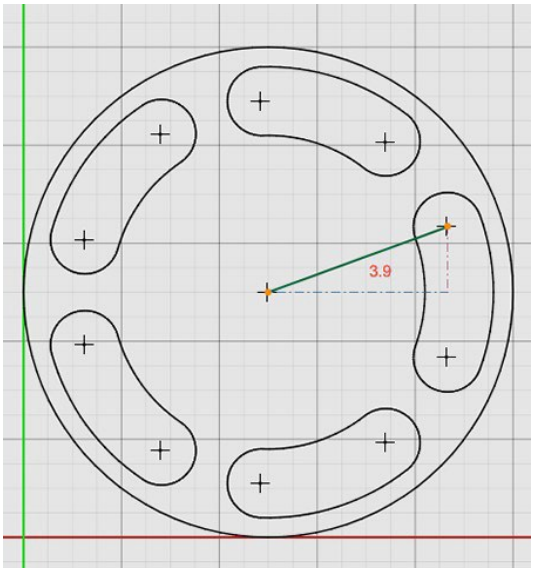
The Point and Angle rotation requires you to select a central rotation point and then enter a rotation amount in degrees. After changes are complete, click the green check mark or press Enter to Accept changes or the red X mark to Cancel changes.

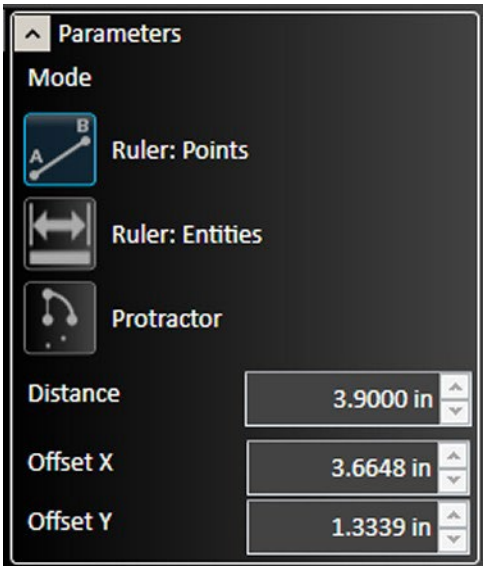




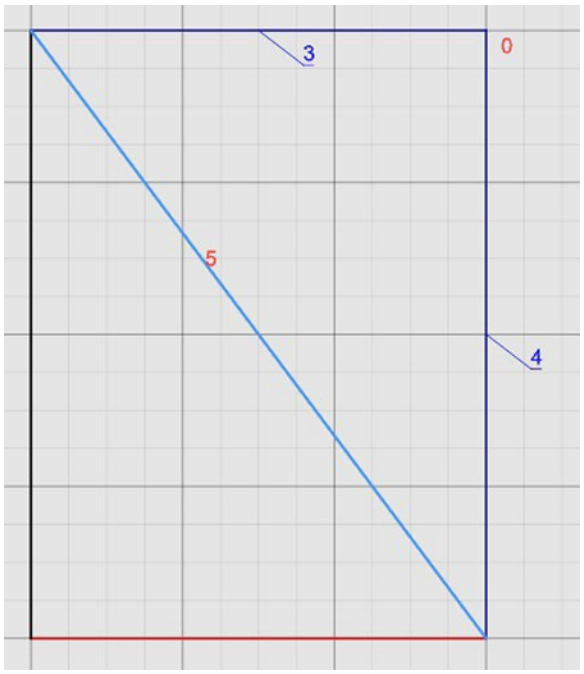
8.23 Measure Entities with Ruler

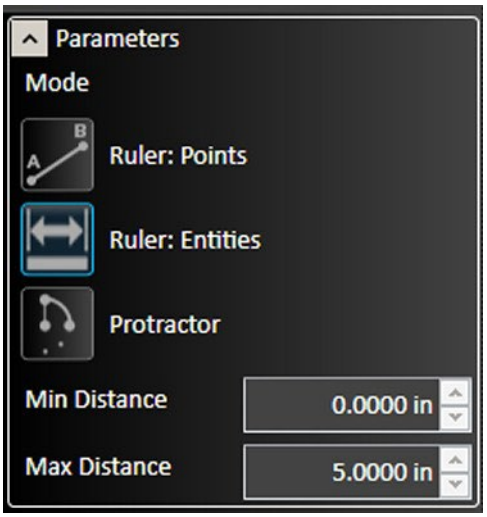
You can measure entities in one of three (3) modes: as a ruler between points or between entities or as a protractor between points on an arc. The tool allows for measurement between any two (2) points and displays offset in X and in Y directions.



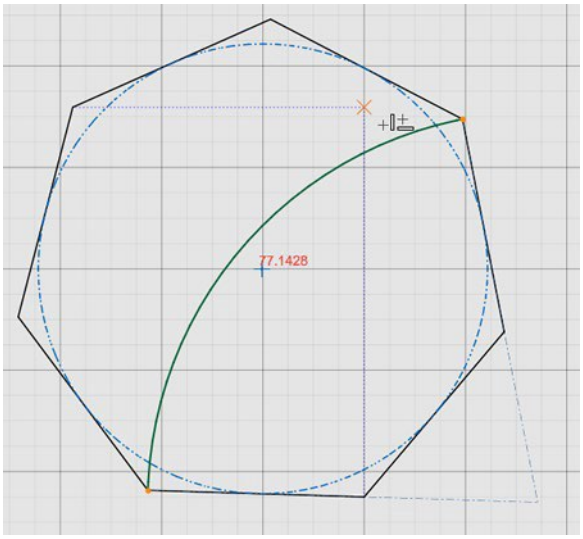


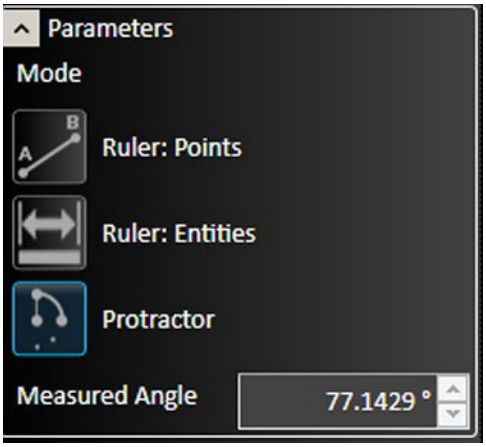
Ruler used to measure between the distance between points center of an arc and radius of an arc. Note the construction lines marking the other sides of the triangle: these represent the Offset X and Offset Y.





Ruler used to measure the distance between two (2) entities: adjacent sides of a rectangle. Note that the Min Distance and Max Distance appear on the drawing, as do the lengths of the two (2) sides.



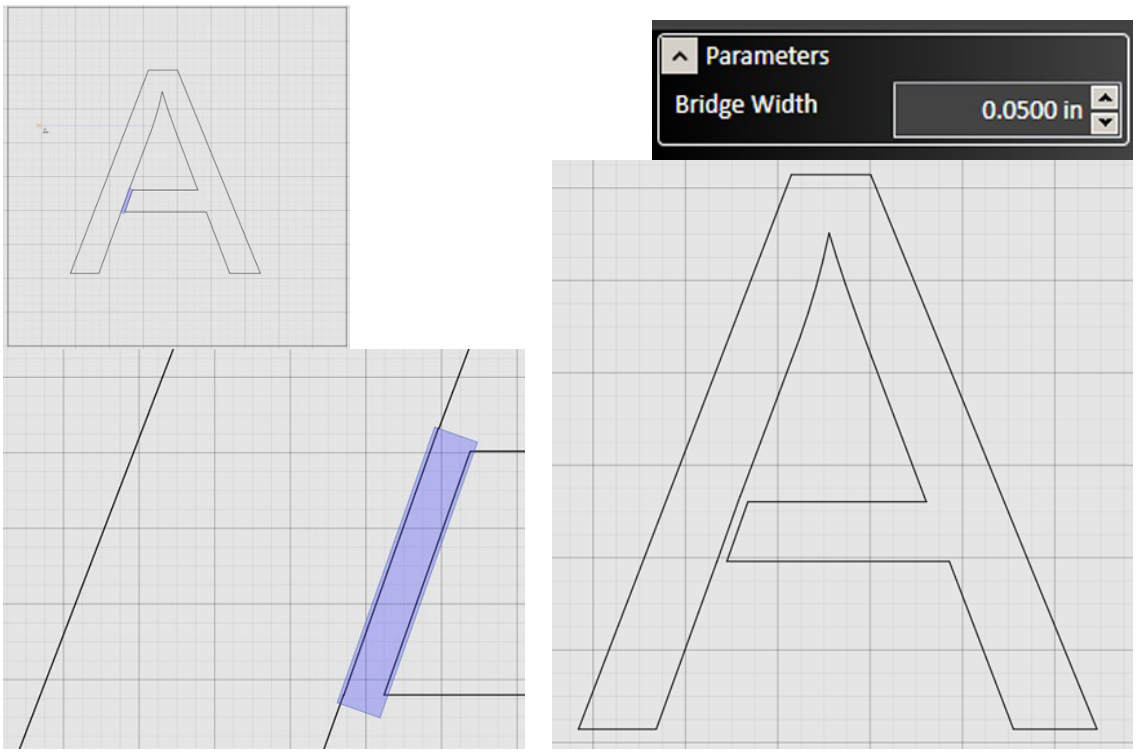


Protractor used to measure the arc and Measured Angle between two (2) points of a polygon. Note that the protractor will not measure an arc or angle if it is grouped with other lines or shapes. You must ungroup the arc to enable the protractor tool.

8.24 Bridge Entities

Connects two (2) 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.

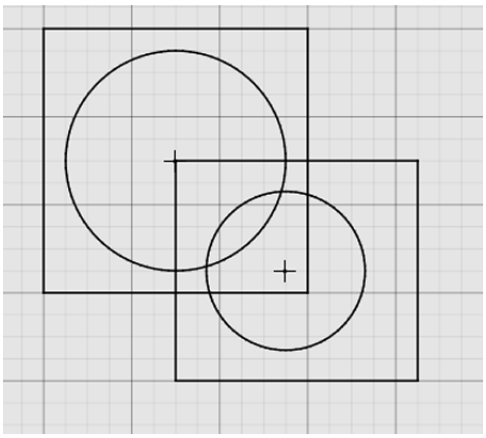
To create a bridge, select Bridge Entities. Use the cursor to draw a line. This line will become the bridge. You can modify the width of the bridge if desired. Select the green checkmark to Accept the change.

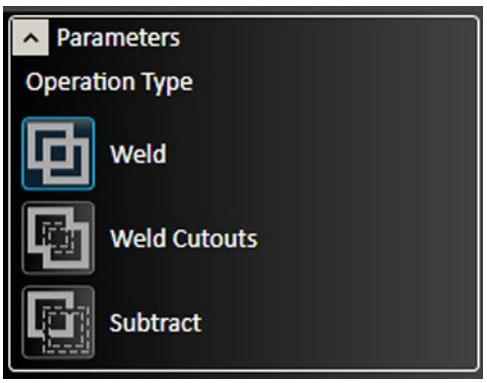


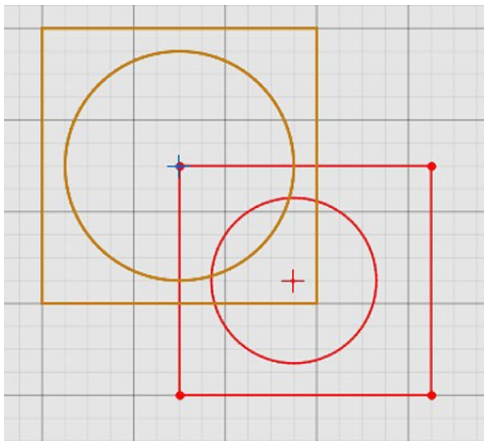
8.25 Perform Boolean Operation on Entities

Perform Boolean Operation can be used to modify one shape by adding or subtracting another shape to it. The three (3) available operations are Weld (combine) solid areas, Weld Cutouts, or Subtract.

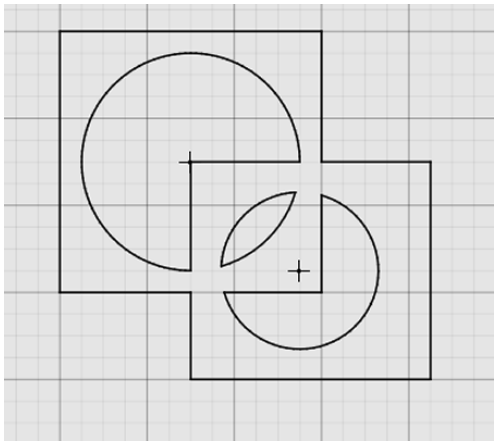
Select Perform Boolean Operation, select the specific operation in the parameters window, then select the objects one at a time, right clicking to confirm each selection. The first object selected will be modified by the second. The example below shows the results of all three (3) operations on the initial of shapes.



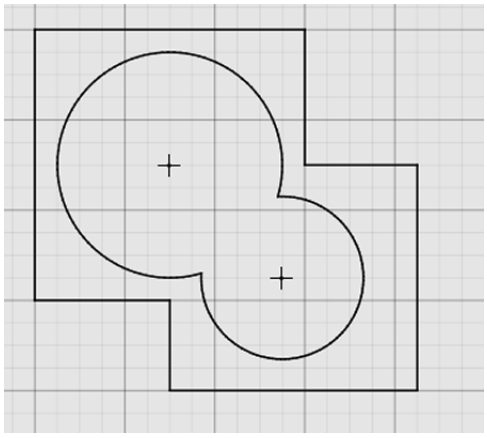




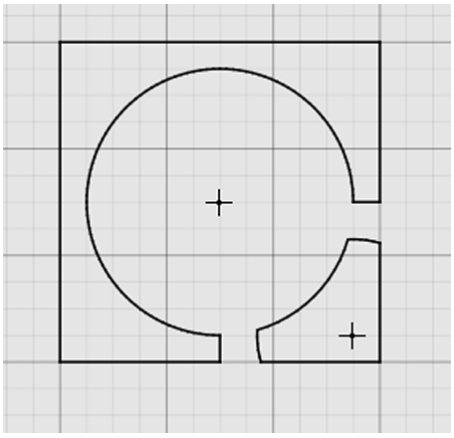
Shapes highlighted to indicate which object is first (gold) and second (red).



The results of a Weld operation.



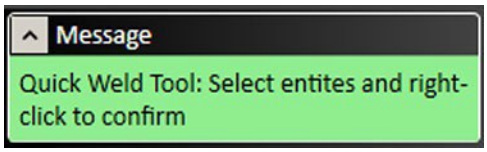
The results of a Weld Cutouts operation.



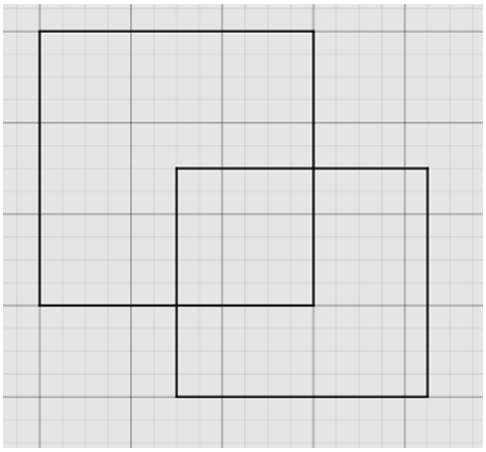
The results of a Subtract operation.

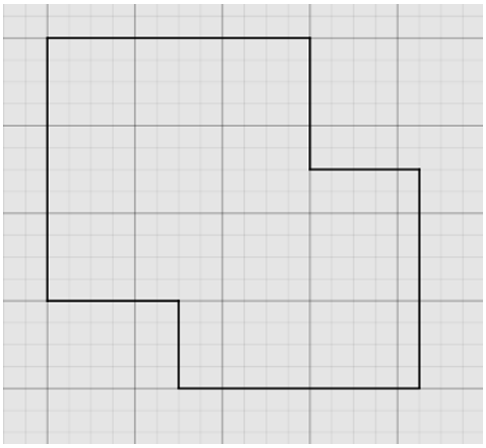
8.26 Weld Selected Entities

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.



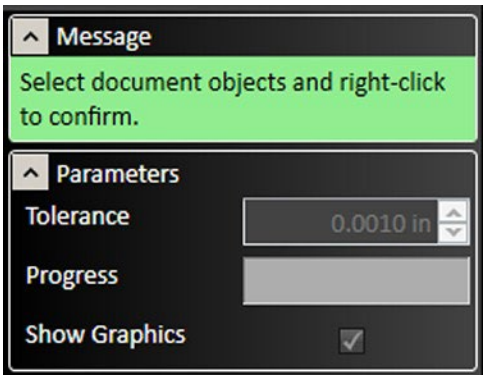


8.27 Arc Fitting Tool

Create arc and line segments for objects that are not already drawing objects, such as imported images.

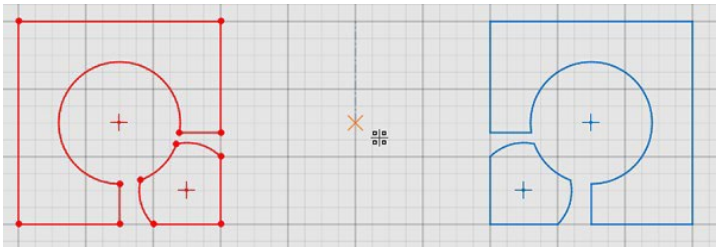
The Tolerance can be set in drawing units. A lower number will increase accuracy. Progress during recalculation is shown in the bar.

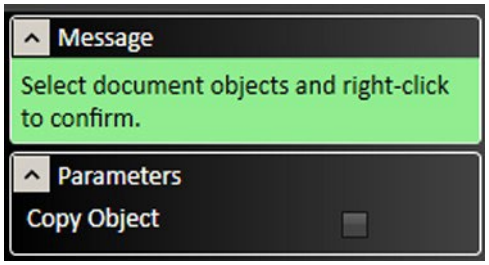
When Show Graphics is selected an outline lines and arcs will be displayed in the drawing window.



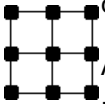
8.28 Mirror Entities

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.





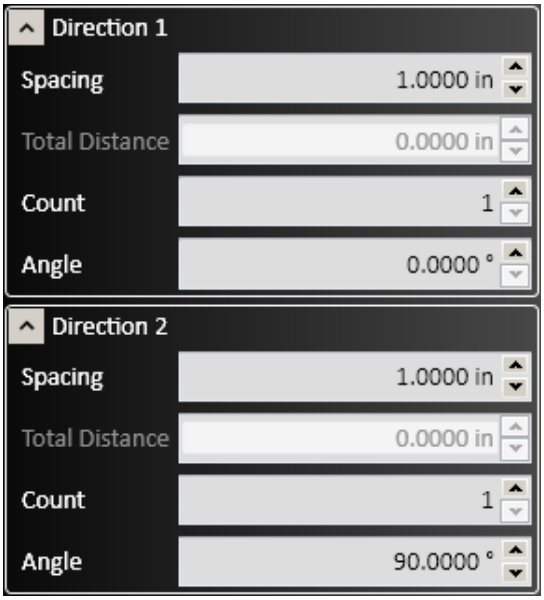
8.29 Linear Pattern Tool

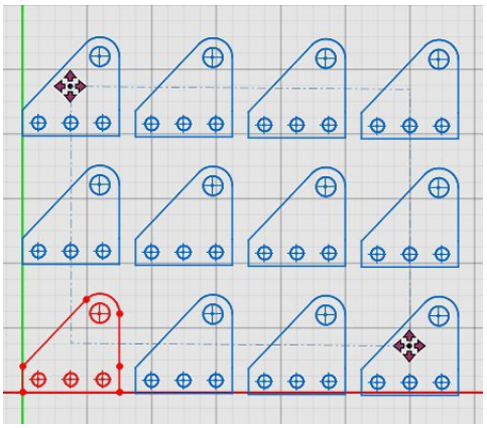


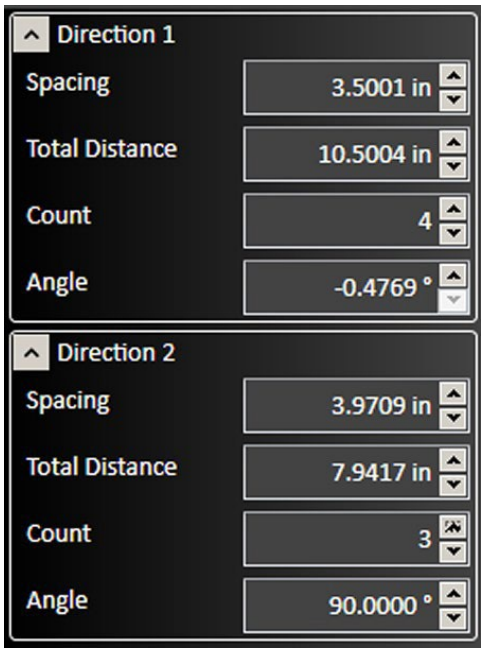
The linear pattern tool duplicates selected features in a grid fashion. Select the feature to be duplicated select the Linear pattern tool then specify parameters.

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 Enter to accept changes or the red X mark to cancel changes.

- Direction 1 represents the horizontal direction.
- Direction 2 represents the vertical directions.
- Spacing sets the distance between copies.
- Total Distance represents the overall distance from the original to the final copy.
- Count indicates the number of copies to generate in each row and column.
- Angle specifies the angular offset from the horizontal and vertical directions.

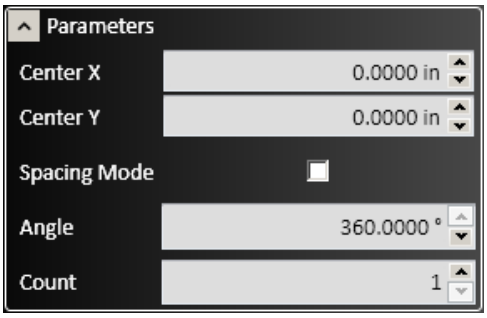




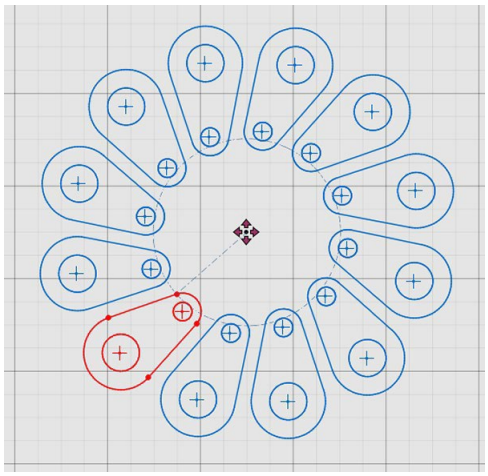


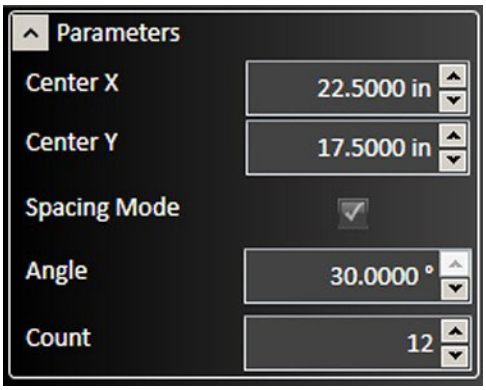
8.30 Circular Pattern Tool

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.

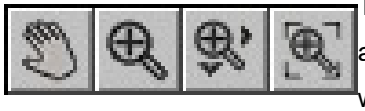


- The Center X and Center Y options specify the point about which the feature will be rotated. Alternately, click and drag the blue center point of the rotation.
- Checking the Spacing Mode box changes the Angle parameter to represent the degree angular spacing between each copy, rather than from the original to the final copy.
- The Angle parameter indicates the angular position of the final copy from the original.
- The Count option determines the number of copies that will be created.





8.31 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.

1. **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.

2. **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.

3. **Zoom Extant**

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.

4. **Zoom to Fit**

Click this button to adjust zoom automatically to best fit the drawing.

8.32 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 already. 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 night clicking in the drawing window. This brings up a contextual menu, including a sub-menu to control snaps.

These are the tools that are available on the Snap tools toolbar at the top right of the workspace:

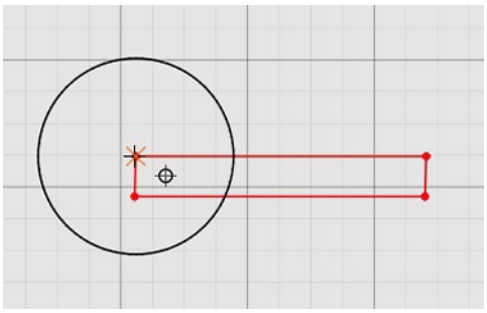
All Snap	Midpoint snap
Center snap	Nearest snap
Endpoint snap	Parallel snap
Grid snap	Perpendicular snap
Horizonal/Vertical snap	Quadrant snap
Intersection snap	Tangent snap

5. All Snap

When All snap is selected, individual snap options may be selected. No snap can be selected when All snap has been deselected.

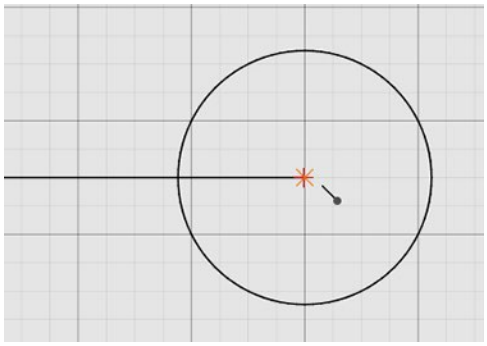
6. Center snap

Features connect to the centers of elements.

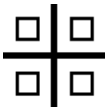


7. **Endpoint snap**

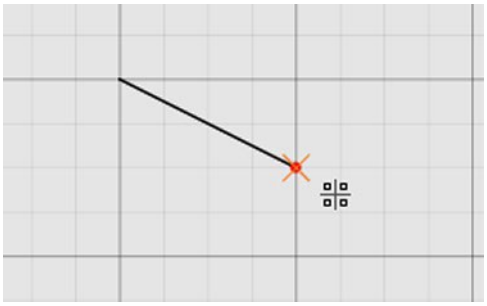
Features connect to the endpoints of elements.



8. **Grid snap**

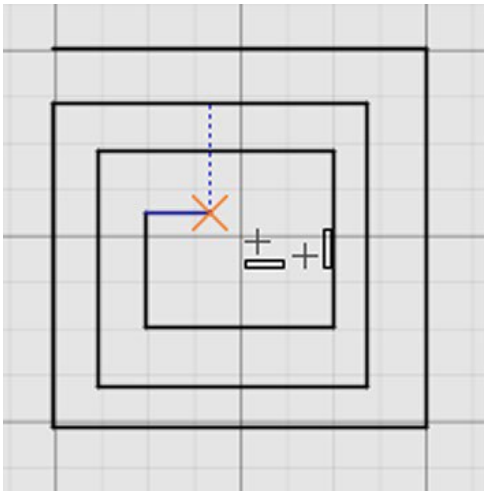


Features connect to points located along the background gridlines.



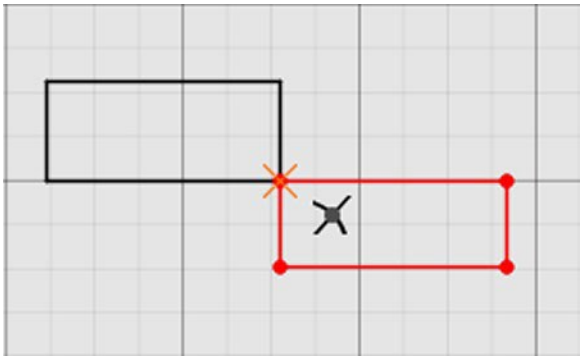
9. **Horizontal/Vertical snap**

Features align vertically and horizontally when pulled into either direction.



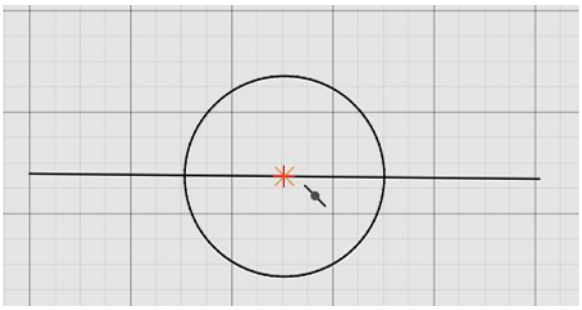
10. Intersection snap

Features connect to the intersection points of elements in the drawing.



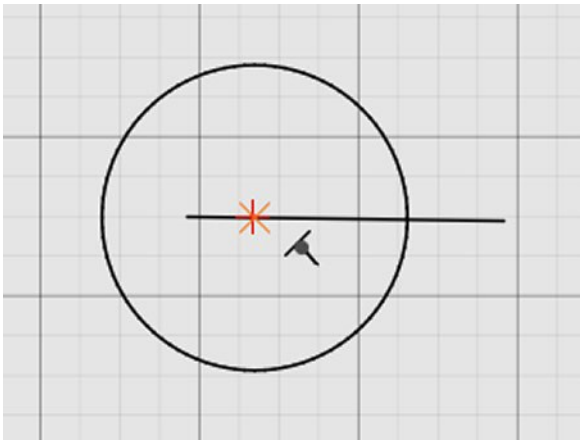
11. Midpoint snap

Features connect to the midpoints of elements.



12. Nearest snap

Features connect to the element closest to the cursor.



13. Parallel snap

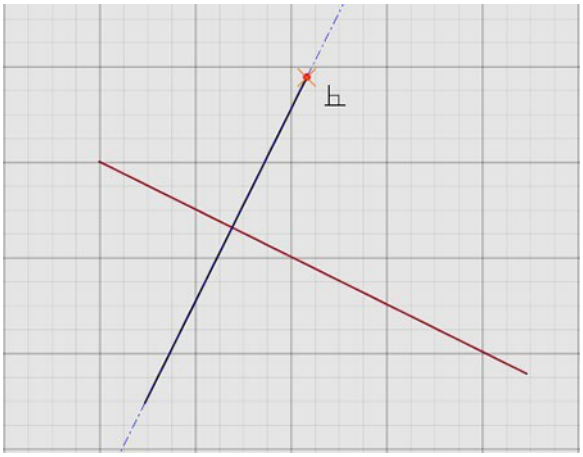
Features snap to a line parallel to the selected element.

Right: a blue dashed construction line indicates that the bottom is parallel to the red highlighted line.

14. Perperpendicular snap

Features snap to a line perpendicular to elements of the drawing.

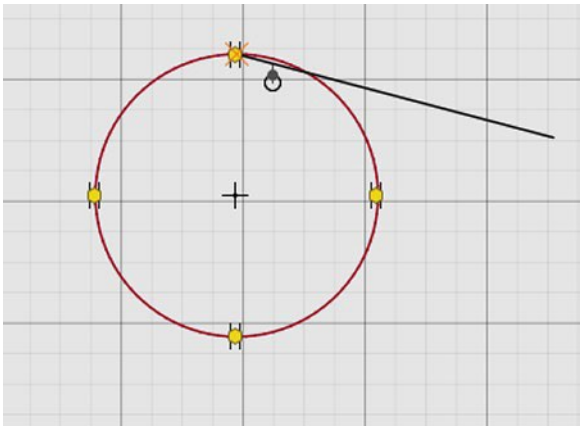
Right: a blue dashed line indicates that the line is perpendicular to the red highlighted line in the drawing.



15. Quadrant snap

Features connect to the topmost, bottommost, leftmost, and rightmost points of elements.

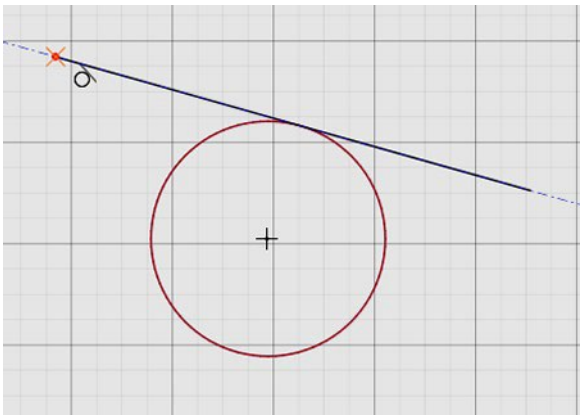
Right: line connected to a point at the top of the circle.



16. Tangent snap

Features snap to a line tangential to elements in the drawing.

Right: a blue dashed construction line indicates that the line is tangential to the circle.

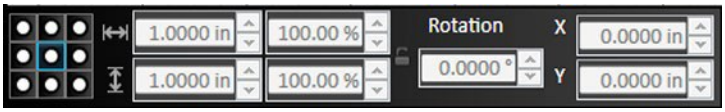


8.33 Free From Stretching and Manipulation Tools

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 the 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.

- Control handle grid
- Horizontal and vertical dimensions
- Rotation
- X and Y position



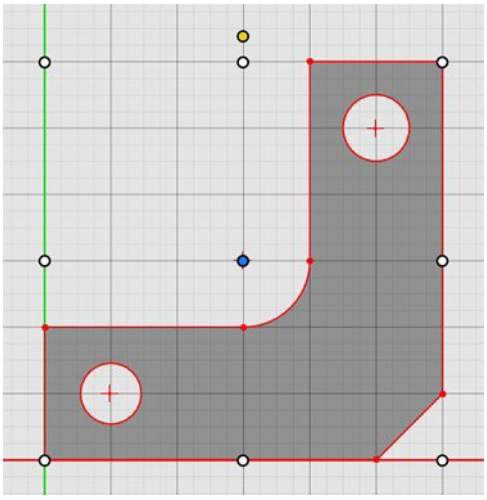
8.34 Control Handle Grid

Selecting an object or objects in the drawing workspace displays a grid of control points.

To move the object, click and drag the blue control point. The blue point is also used for rotation and to determine the position of the object.

The blue control point is selected using the grid tool. The selected point is shown in blue in the drawing workspace (right) and is highlighted in blue on the grid (left).

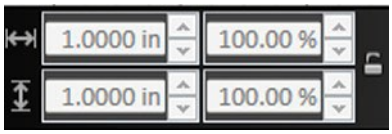
To scale the object, click and drag any white control joint.



To rotate the object, click and drag the yellow control point.

8.35 Horizontal and Vertical Dimensions

The horizontal and vertical dimensions of the area under control are shown in two (2) ways: as measured in units on the drawing workspace grid and as a percentage.



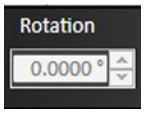
Changes made by moving the control points are reflected in the fields. Changes can also be made directly to each field.

Click the lock icon to lock the aspect ratio.

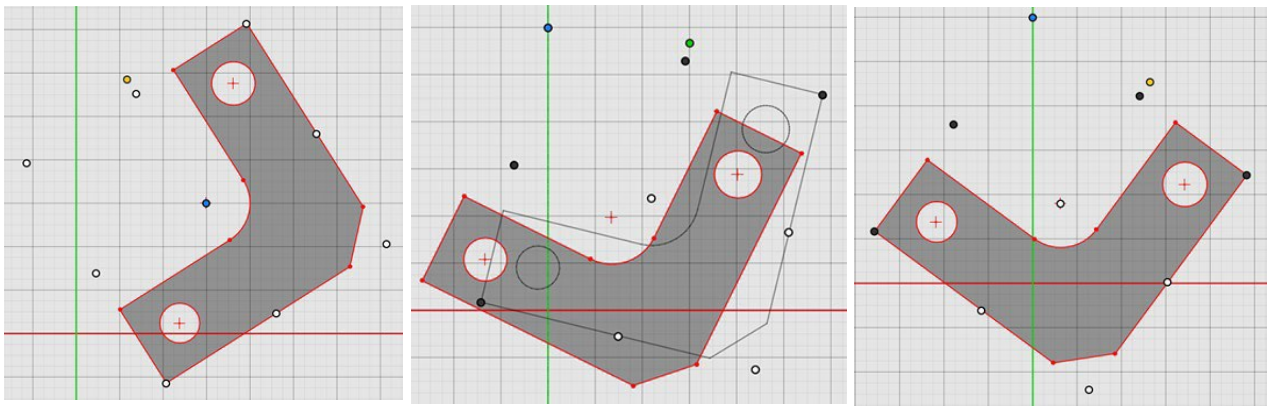
8.36 Rotation

The selected (blue) control point defines the center of rotation.

The degree of rotation is displayed in the field. Changes can also be made directly to the field.



Below, the bracket is shown rotating around the middle (left) and upper right (center, right) control points. Note that the outline of the image is visible (center) during a move to show the new position.



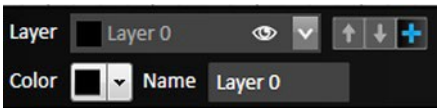
8.37 X and Y Position

The X and Y positions of the area under control, measured from the selected (blue) control point are shown in the fields.

Changes can also be made directly to each field.

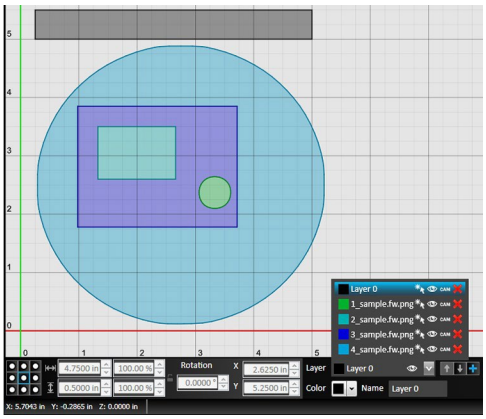
8.38 Layers

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 treated in Flash cut CAM.



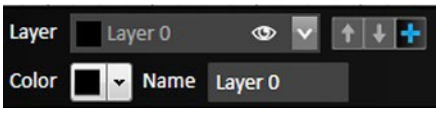
- Layers
- Color
- Name

In the example below, an image has been imported using Import color image. Each shape now occupies a different, named, color-coded layer. The layers are stacked, with layer 0 on top, and layer 4 on the bottom. Note that the layer colors are also used by the part shading tool.

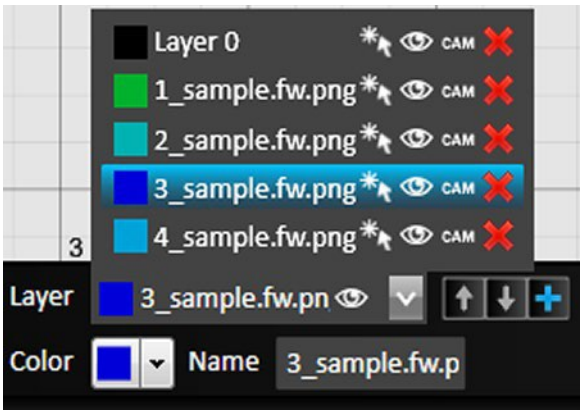


8.39 Layer

The information about the active layer is always displayed in the toolbar. When the layer menu is displayed additional details are shown about all layers, and the color, name, and visibility of the active layer are always displayed in the toolbar.



When the arrow icon is clicked, more information about all layers is displayed. Note that active layer is highlighted.



Four (4) clickable icons are visible:

- Select all objects from this layer.
- Show or hide this layer.
- Use this layer in CAM.
- Remove layer.

Use the arrows to move the layer up or down in the stack.

1. **Move Layer up/down**

   Use the arrow icons to move the layer up or down in the stack.

2. **Add layer**

   Use the plus sign icon to add a new layer.

3. **Color**

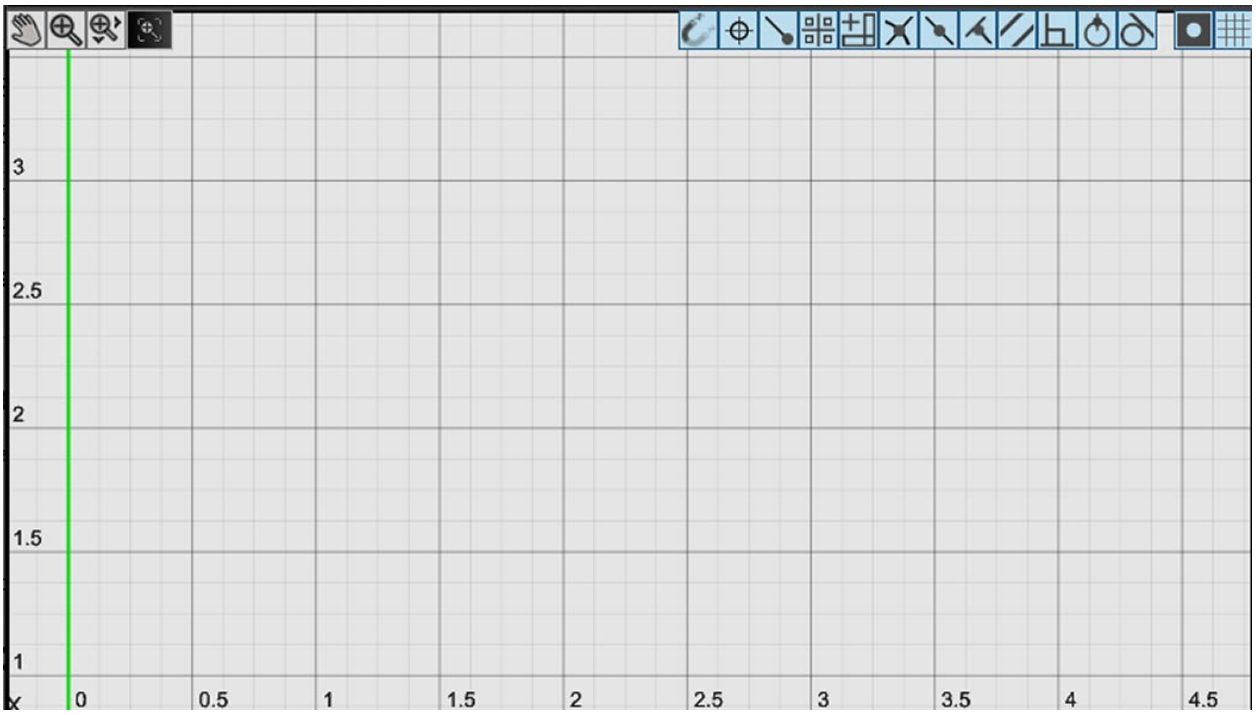
   The color of the active layer can be changed here. Clicking the arrow brings up a preset menu of colors. More colors can also be defined, using the Windows color tool.

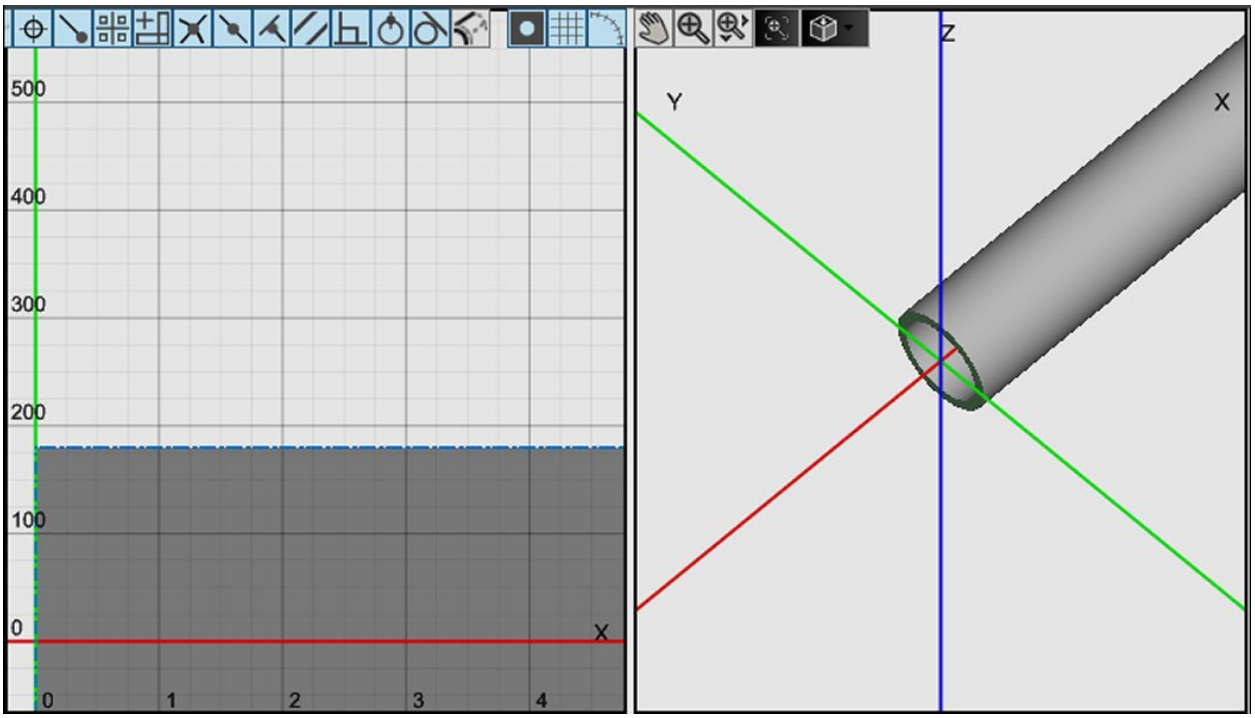
4. **Name**

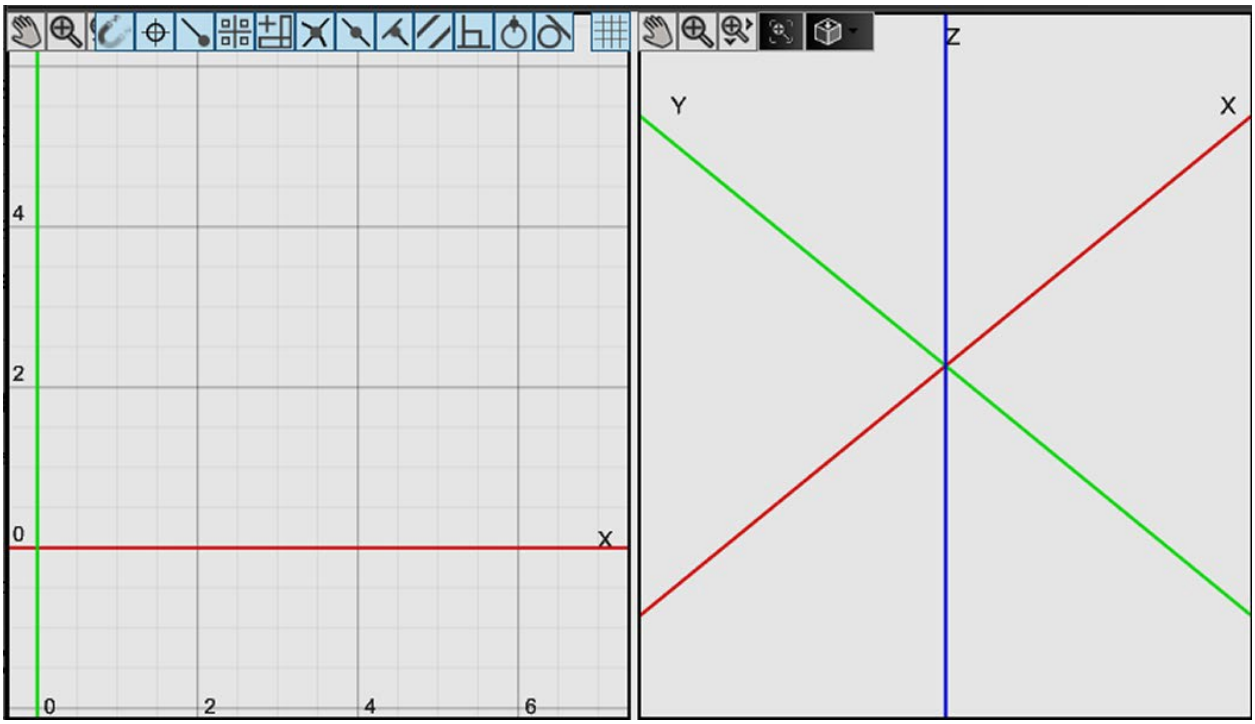
   The name of the active layer can be edited in this field.

5. **Drawing workspace**

   In FlashCut CAD, the drawing workspace is different for each type of drawing. Examples of a 2D part drawing, 3D part drawing, and Pipe/Tube part drawing are shown below. The default workspace when opening the program is for a 2D part drawing.


3
2.5
2
1.5
1
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5


500
400
300
200
100
0
X
0
1
2
3
4
Z
Y
X



6. **Viewport Tools**

   The drawing workspace for a 3D part drawings or a Pipe/Tube part drawing displays an additional right-hand viewport that displays a 3-Axix view of the drawing. Additional tools are present in the upper left corner of the viewport.

7. **Pan and Zoom Tools**

   The Pan and Zoom Tools function in the same way as those in the main drawing workspace.

8. **Orientation Tool**

   The right-hand view port can be set to display any face of a three-dimensional object or an isometric view (default).

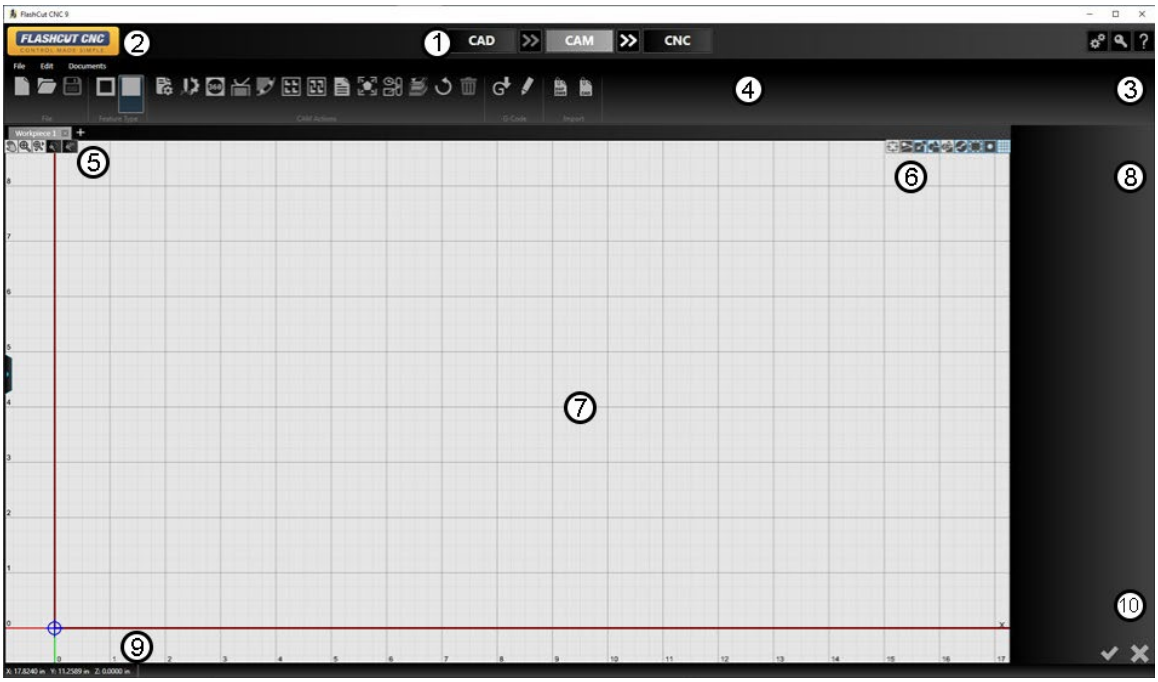
   Position the cursor over the orientation tool icon to display and select either a specific face of an orthographic view (front, top, left, etc.) or an isometric view.

NOTES:

9.0 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.



1. Tabs
2. Menu Bar
   Menu bar print commands
3. Configuration, License, and Help Buttons
4. Ribbon
5. Pan and zoom tools
6. Display options tools Part shading and Grid Tools
7. Drawing workspace
8. Parameters area
9. Status bar
10. Accept/Cancel

9.1 Menu Bar Print Commands

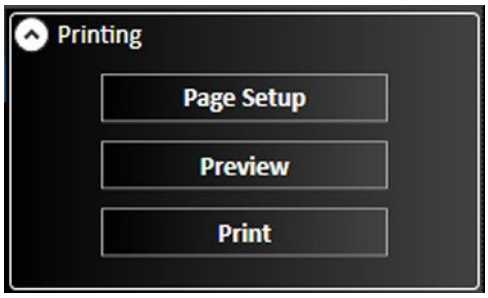
9.1.1 Print

FlashCut CAM adds a set of print commands to the File menu.

- Print Labels
- Print Nest

9.1.2 Print Labels

The prints labels command can be used to create labels displaying part names and part numbers.



1. **Page Setup**

   Brings up a Windows page setup window, allowing you to set page size, layout, paper source, and all margins.

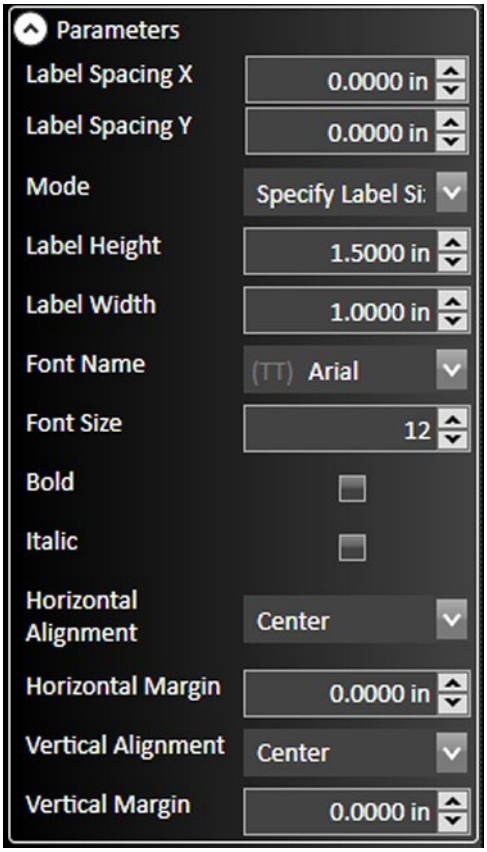
2. **Preview**

   Brings up a age preview window. Displays multiple page at varying resolutions and layouts.

3. **Print**

   Brings up a Window printer selection window.

9.1.3 Label Spacing X/Y



1. **Mode**

   Select either Specify Label Size or Specify Grid Dimensions.

2. **Label Height/Width**

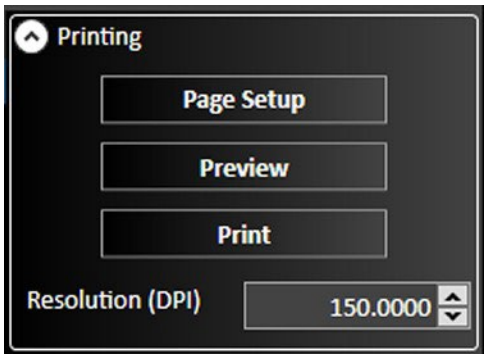
   Label size can be set by X/Y dimensions in system units.

3. **Columns/Rows Number**

   Grid dimensions can be set by assigning the number of columns and rows per sheet of labels.

4. **Font Attributes**

Select from available system fonts and font sizes. Add font styles using checkboxes for bold and italic.



5. **Horizontal/Vertical Alignment**

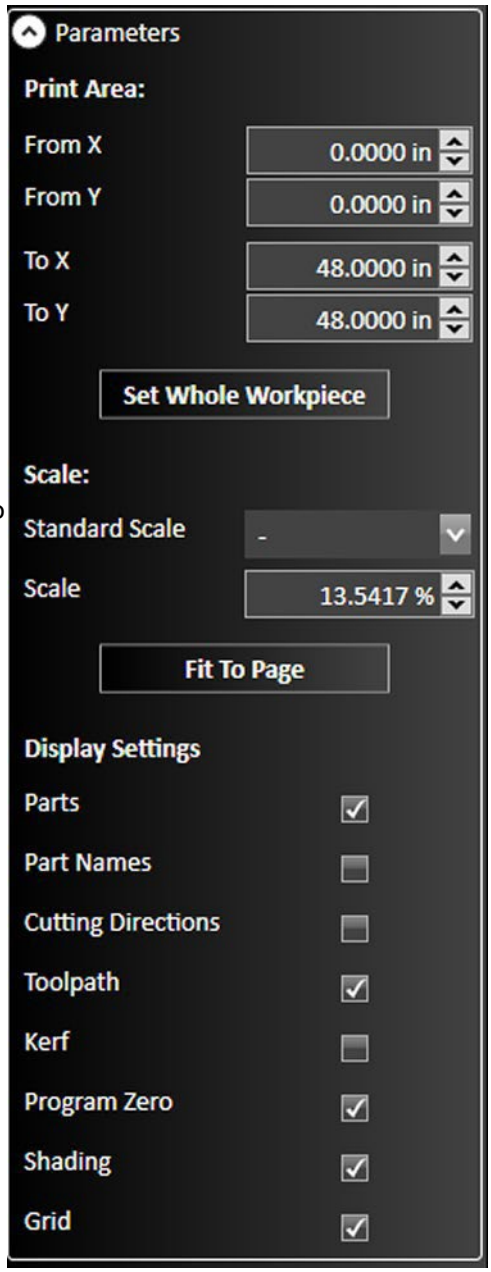
   Set label text alignment: Left, Center, Right, and Top, Center, Bottom.

6. **Horizontal/Vertical Margin**

   Set margins for label text.

9.1.4 Print Nest

Print a drawing showing the active workspace. If either or are defined, the nesting pattern will be shown.



1. **Page Setup, Preview, Print**
   Same as for Print Labels.

2. **Resolution (DPI)**
   Set value up to 600 DPI. Default is 150 DPI.

3. **Print Area Coordinates**
   Area to be printed can be set by defining the four (4) corners of a rectangle: From X, To X, From Y, and to Y.

4. **Set Whole Workpiece**
   Select button to include the whole workpiece in the drawing.

5. **Standard Scale**
   Select a scale from the drop-down menu: 1:50 to 10:1.

6. **Scale**
   Set scale of drawing as a percentage of actual size.

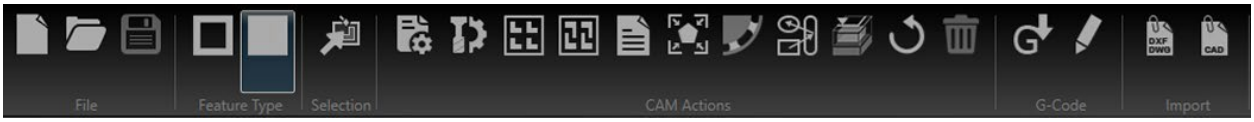
7. **Fit to Page**
   Fit drawing to a single page. Automatically calculates scale.

8. **Display Settings**

Select which features of the drawing workspace are visible in the print. See Display options tools for definition.

- Parts
- Part Names
- Cutting Directions
- Toolpath
- Kerf
- Program Zero
- Shading
- Grid

9.2 Ribbon



The FlashCut CAM ribbon has six (6) types of command icons:

- File Tools
- Feature Type
- Selection
- CAM Actions
- G-Code
- Import

9.3 File Tools

There are two (2) file tools on the ribbon:

- Open (Ctrl+O)

- Save (Ctrl+S)

9.3.1 Open (Ctrl+O)

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

9.3.2 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 and destination. FlashCut saves drawings to the CAD/CAM *.cadcam) format. These files represent CAD drawings readable by FlashCut CAD version 5 or later. This command is unavailable when no recent changes have been made.

9.3.3 Feature Type

This pair of settings controls whether the top level of the drawing, indicated by the outermost line, is considered a part or a cut out. This setting will change which sections of the drawing are considered scrap and which part is to be saved. The placement of kerf lines, lead ins, and corners is determined by this setting. For example, when marking a circular cut in the workpiece, is the intention to make a disc, or to make a circular hole? The former is a part; the latter is a cutout. The behavior of nested closed shapes is also determined by this setting.

Changing from one to the other will reset the history of the Undo command.

9.3.4 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.

9.3.5 Make Top Level Feature a Part

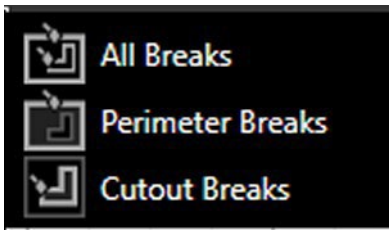
If this option is selected, the outermost level of the drawing will be understood as a part. For example, a simple shape (circle, rectangle, etc.) will be cut so as to preserve the material inside the line defining the boundary of the shape. The lead in will begin outside the boundary defined by the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

NOTE

This option is selected by default.

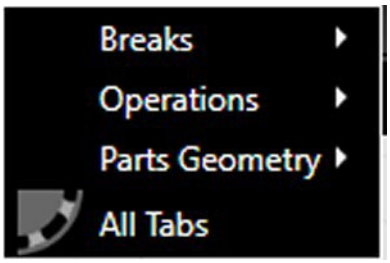
9.3.6 Selection

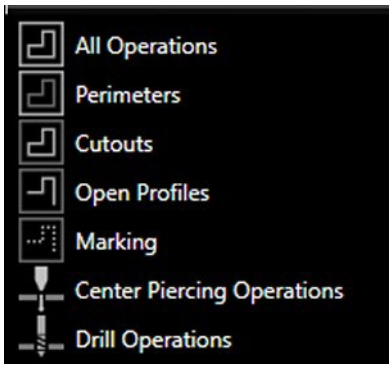
The Selection icon is visible if the installed Fab Head supports it. As present, only Spindle Fab Head support Selection.



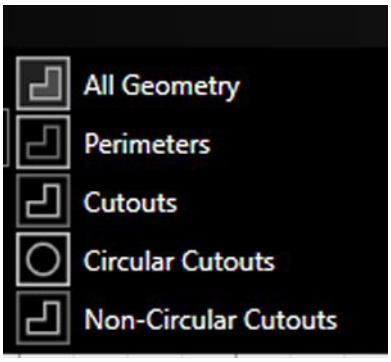
The Selection interface allows you to select multiple drawing attributes so that parameters affecting types of features or operations can be modified more efficiently.

When you click the Selection icon, FlashCut pops up the following choices for selection options:





Submenus for Breaks, Operations, and Parts Geometry are shown to the right.



9.4 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.

Some icons are only visible for specific types of Fab Heads. For example, Smart350™ settings are only visible if licensed for a plasma Fab Head.

These are the tools that are available on the CAM actions portion of the ribbon:

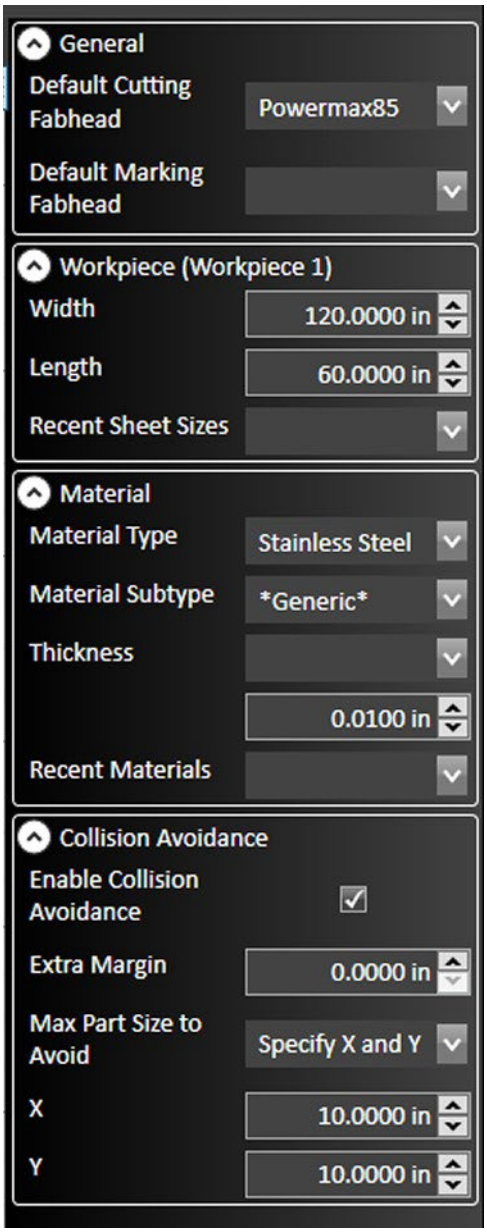
- Project Settings
- Plasma Settings
- Oxyfuel Settings
- Waterjet Settings
- Laser Settings
- Marker Settings
- Spindle Settings
- Knife Settings
- Change Project Break Settings
- Change Project Tab Settings.
- Grid Nesting
- TrueShape Nesting
- Nesting Information
- Fit workpiece to the parts

If a Fab Head has been configured, its icon will appear on the CAM ribbon. For more information, see Configuring FlashCut.

- Sequence Tool
- Simulation Tool
- CAM Reset
- Delete

9.5 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.



9.6 General

1. **Default Cutting Fabhead**

   Select the default cutting fabhead. Multiple fabheads can be defined in the Configuration interface.

2. **Default Marking Fabhead**

   Select the default marking fabhead.

9.7 Workpiece

1. **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.

2. **Length**

   Specify the length 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.

3. **Recent Sheet Sizes**

   Select the sheet size from a list of recently selected material types.

9.8 Material

1. **Material Type**

   Specify the material type and subtype. The parameters will automatically adjust to match your selection.

2. **Thickness**

   Specify the thickness of your material. The parameters will automatically adjust to match your selection.

3. **Recent Materials**

   Select from a drop-down list containing recently selected materials.

9.9 Collision Avoidance

1. **Enable Collision Avoidance**

   Helps prevent collisions with parts already cut. When enabled, a larger margin is added to the Fab Head's path during rapid moves, letting it take corners around parts that could tip up. The parameters can be adjusted to ignore large parts that never tip.

2. **Extra Margin**

   Defines margin to add to path.

3. **Max Part Size to Avoid**

   May be defined by the part's X or Y dimensions, or by both together.

9.10 Change Project Break Settings

Use the Project Settings to control how the installed Fab Head handles default values for break placement and lead definition. Note that individual breaks can be overridden manually. See Lead in/out settings for a general discussion of breaks in the drawing workspace.

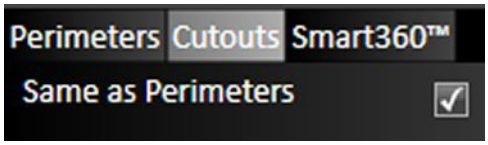
9.10.1 Perimeters

Select this tab to configure breaks that are on the outsiders of parts. When the toolpath is offset to the outside of the part, the break on this toolpath is defined as a perimeter break.

9.10.2 Cutouts

Select this tab to configure breaks that are on the insides of cutouts. When the toolpath is offset to the inside of a feature or a cutout, the break is defined as a cutout break.

Select Same as Perimeters to apply all break settings defined in Perimeters. If unchecked, settings can be configured separately.

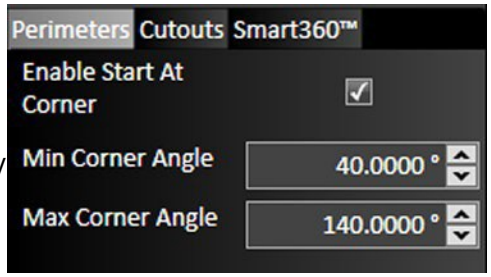


9.10.3 Smart 360™

If Smart 360 is enabled for the Fab Head, break setting to use with it can be defined here.

9.10.4 Enable Start at Corner

If selected, a section labeled Leads for Corners is visible. It displays the same options as Lead for Non-Corners below.



Min/Max Corner Angle – This range of angles will specify whether or not a break will be placed at a corner or not. If the software finds an angle withing range, it will place the break at the vertex of that angle.

If no angles exist that fall in this range then the break is placed at the midpoint of the longest line segment. If no acceptable angles exist and no line segments exist, the break will be placed on an arc nearest the origin.

9.11 Leads for Non-Corners

Sets the default pattern for leads.

9.11.1 Overburn

Set distance. To ensure a clean cut, you may set the torch to continue along the same toolpath. Lead outlines appear after the overburn.

9.11.2 Turn Off Fab Head Early

If desired, the Fab Head can be turned off at different point during a cut.

- Toolpath Completion

  Fab Head remains on until toolpath is done.

- Feature Completion

  Fab Head turned off at the end of the feature.

- Before Feature Completion

  Fab Head Off Distance field displayed.

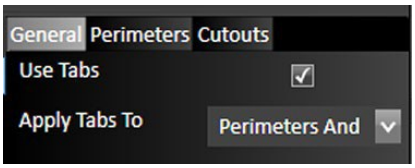
After Feature Completion

- Fab Head Off Distance field displayed.
- Arc creates an arc tangent to the feature. If you choose Arc, the Arc Radius and Length Type options are displayed.

  Length Type options – Length Fixed, Angle Fixed, or Based on Sheet Thickness
- Line results in a straight line. If you choose Line, the Line Length and Line Angle options are displayed.
- None – No lead will be used.

9.12 Change Project Tab Settings

Tabs are used to keep the part attached to the workpiece.



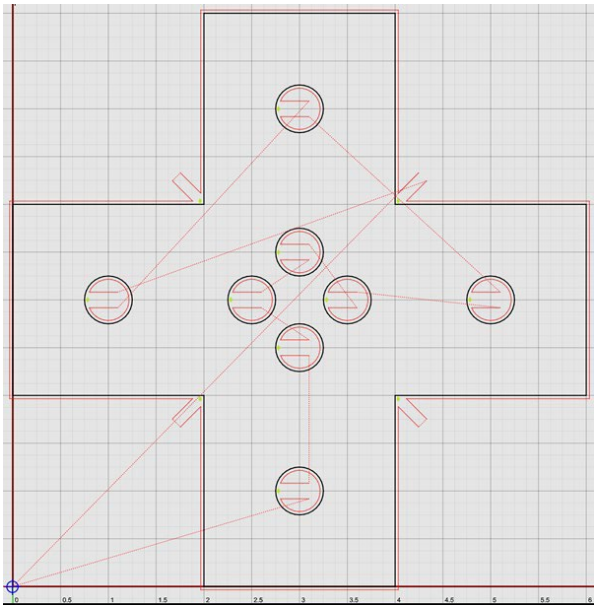
1. **Use Tabs**
   When selected, tabs will be included in the toolpath for the drawing.
2. **Apply Tabs**
   Tab settings can be configured for perimeters and cutouts, either separately or together. Three (3) labeled tabs enable detailed configuration.

In this drawing, four (4) perimeter tabs have been used to keep the part connected to the workpiece. The tabs appear at the right-angle corners formed by the arms of the cross.

In addition, one tab has been defined for each cutout, keeping the disks inside each hole attached to the part.

Same as Perimeters

Settings for Cutouts can be defined separately or linked to the settings for perimeters.



3. **Stich Tab**

   When selected, tabs are "stitches" of uncut material along the line of the toolpath.

5. **First Tab Shape Tab Width**

   Define tab width.

6. **Lead Line Length**

   Define tab length.

7. **Lead Line Angle**

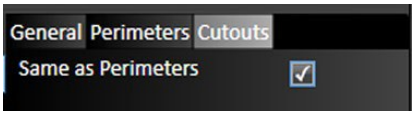
   Define angle of line joining the edge of the tab to the part.

8. **Tabs Placing Method**

   a. Specify Spacing places tabs by defining the space between them.

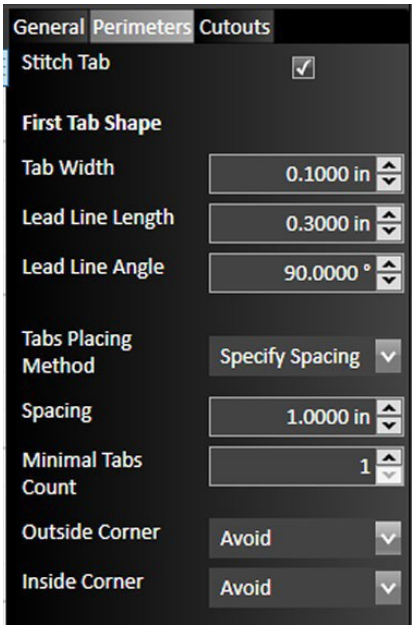
      i. Spacing

      ii. Minimal Tabs Count



   b. Specify number places tabs by setting the absolute number of tabs.

      i. Count of Tabs



9.12.1 Outside/Inside Corner

Select whether FlashCut CAM should Avoid, Consider, or Prefer corners when placing tabs.

9.13 Plasma Settings

If a plasma Fab Head has been configured, its settings can be adjusted here.

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various plasma torches. 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 the settings in the *.setup file. 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. See Configuring FlashCut for more details.

1. **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 by input manually.

   Quality Level

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

2. **Lookup/Reset to Default**

   Return any changed fields to values from cut charts.

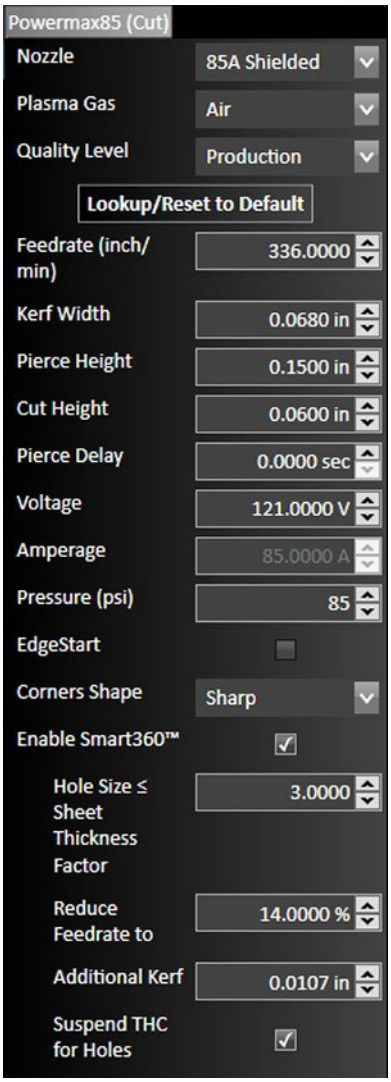
3. **Feedrate**

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

4. **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 ½ the kerf width for parts and automatically offset inside by ½ the kerf width for cutouts.

   Pierce Height



Specify the height at which the torch will automatically 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.

5. **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.

6. **Pierce Delay**

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

7. **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 is dependent on the material type, thickness, torch settings, and other parameters.

   When sampling is enabled, the system sill 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.

8. **Amperage**

   Specifies for operating amperage of the torch.

9. **Pressure**

   Specifies the air pressure of the torch in psi.

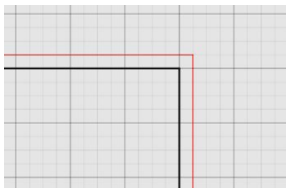
10. **Edge Start**

    When checked, indicates that the cut will start at the edge of the workpiece.

11. **Corners Shape**

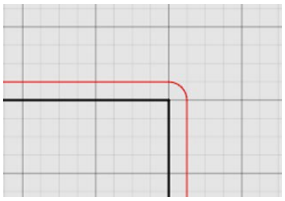
    Toggle whether the owners on the toolpath will be sharp or sounded. Different types of corners will result in more precise cuts depending on the qualities of the Fab Head and the material.

Changes to the corner settings may take a short time to render the toolpath.



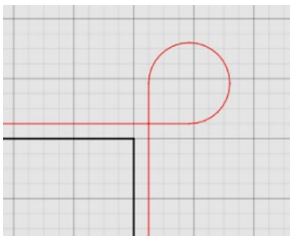
- Sharp Corners

  The toolpath follows the same angle as the part, maintaining a consistent distance from the edge.



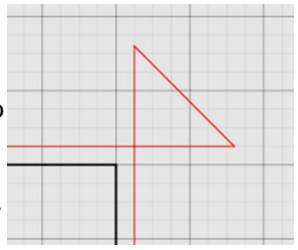
- Rounded Corners

  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.

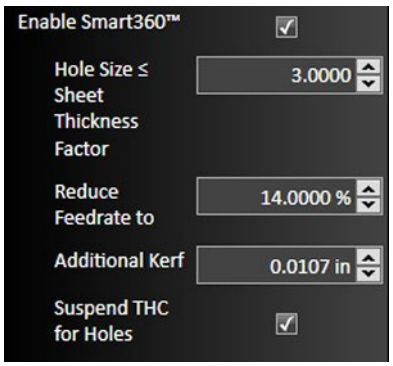


- Triangular Loops

  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.

12. Enable Smart360™

Creates high accuracy holes and small cutouts by manipulating torch, kerf, and feedrate settings. Available with some Fab Heads.



- Hole Size ≤ Sheet Thickness Factor
- Reduce Feedrate
- Additional Kerf
- Suspend THC for Holes

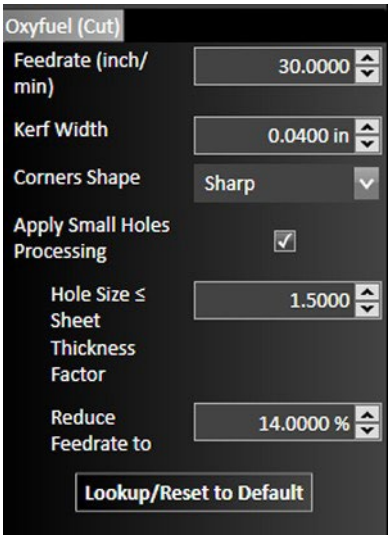
9.14 Oxyfuel Settings

If an oxyfuel Fab Head has been configured, its settings can be adjusted here.

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various Fab Heads. 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 the settings in the *.setup file. 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. See Configuring FlashCut for more details.



1. **Feedrate**

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

2. **Kerf Width**

   Input the width of the kerf of the oxyfuel cutter. 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 ½ the kerf width for parts and automatically offset inside by ½ the kerf width for cutouts.

3. **Corners Shape**

   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.

   For an example, see Corners in the section on Plasma Fab Heads, above.

4. **Apply Small Holes Processing**

   When selected, enables cleaner cutting of small holes.

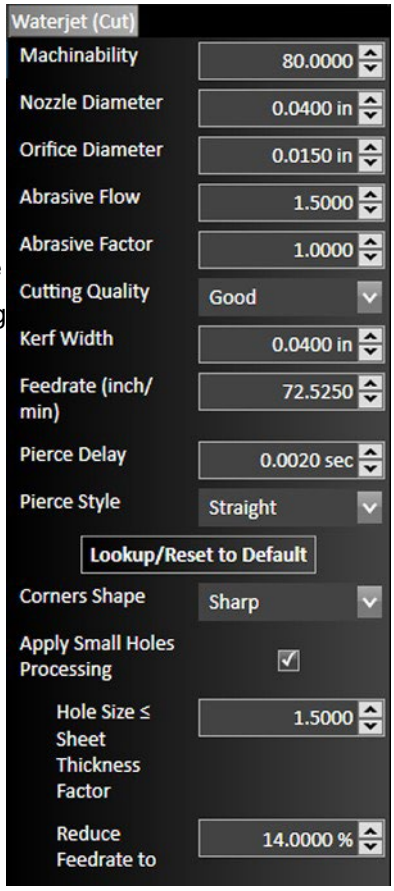
   - Hole Size ≤ Sheet Thickness Factor
   - Reduce Feedrate to n%

5. **Lookup/Reset to Default**

   Return any changed fields to value from cut charts.

9.15 Waterjet Settings

If a waterjet Fab Head has been configured, its settings can be adjusted here.



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 the settings in the *.setup file. 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. See Configuring FlashCut for more details.

1. **Machinability**

   This parameter is specific to the material to be cut and is populated from the cut charts for the Fab Head.

2. **Nozzle Diameter**

   Nozzle diameter for the cutting head.

3. **Orifice Diameter**

   Orifice diameter for the cutting head.

4. **Abrasive Flow**

   Flow rate for the abrasive compound.

5. **Cutting Quality**

   Low, fair, average, good, or excellent.

6. **Kerf Width**

   Input the width of the kerf of the waterjet.

7. **Feedrate**

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

8. **Pierce Delay**

   Delay in seconds before a pierce operation.

9. **Pierce Style**

   Options are Straight, Wiggle, or Circular.

10. **Lookup/Reset to Default**

    Return any changed fields to values from cut charts.

11. **Lookup/Reset to Default**

    Return any changed fields to values from cut charts.

12. **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.

    For an example, see Corners in the section on Phasma Fab Heads, above.

13. **Apply Small Holes Processing**

    When selected, enables cleaner cutting of small holes.

    - Hole Size ≤ Sheet Thickness Factor
    - Reduce Feedrate to n%

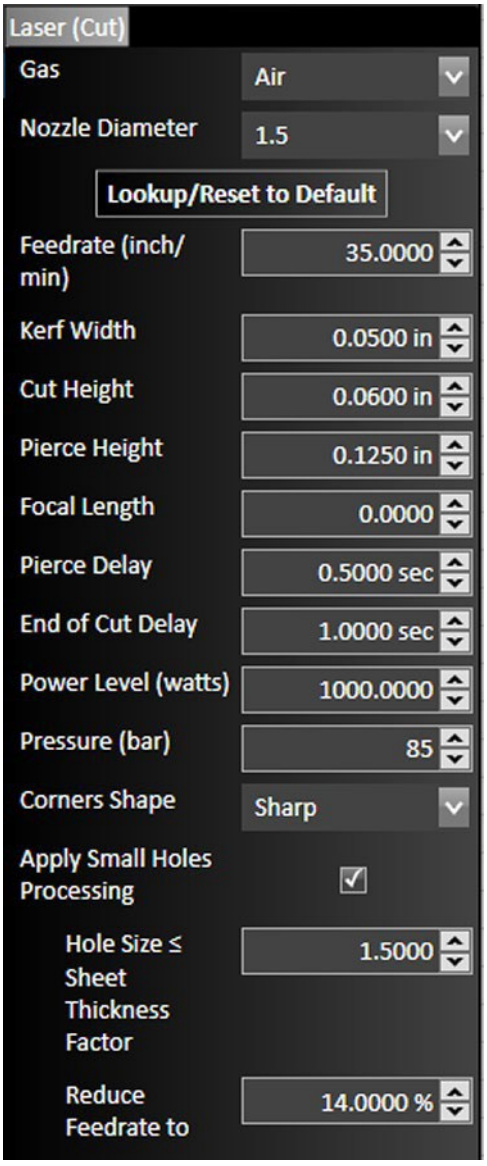
10.0 Laser Settings

If a laser cutter Fab Head has been configured, its settings can be adjusted here.

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various Fab Heads. 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 filed. After specifications are complete, you may either Accept or Cancel the changes.

The values in the parameters are derived from the settings in the *.setup file. 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. See Configuring FlashCut for more details.



1. **Gas**

   Type of gas used in the cutting head.

2. **Nozzle Diameter**

   Nozzle diameter for the cutting head.

3. **Lookup/Reset to Default**

   Return any changed fields to values from cut charts.

4. **Feedrate**

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

5. **Kerf Width**

   Input the width of the kerf of the laser's path. This will determine the thickness of the cut and toolpath will change to reflect the new size. The toolpath is automatically offset outside by ½ the kerf width for parts and automatically offset inside by ½ the kerf width for cutouts.

6. **Cut Height**

   Specify the height at which the laser will cut the part along its toolpath.

7. **Focal Length**

   Focusing distance of cutting head.

8. **Pierce Delay**

   Specify the time between the command for the laser to ignite and the motion of the machine. The delay allows the laser time to pierce completely through the material prior to any other machine motion.

9. **End of Cut Delay**

   Specifies the delay at the end of the cut. The purpose of the delay is to make sure the cut is complete.

10. **Power Level**

    Power level (watts) of the laser cutting head.

11. **Pressure**

    Pressure of the Fab Head's assist gas jet.

12. **Corners Shape**

    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.

    For an example, see Corners in section on Plasma Fab Heads, above.

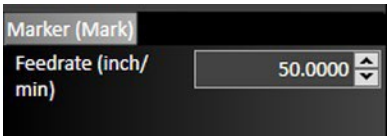
13. **Apply Small Holes Processing**

    When selected, enables cleaner cutting of small holes.

    - Hole Size ≤ Sheet Thickness Factor
    - Reduce Feedrate to n%

10.1 Marker Settings

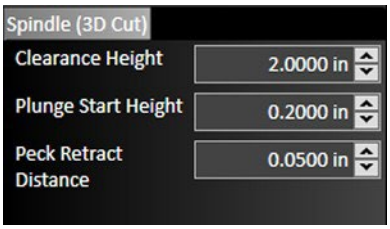
If a marker has been configured, the Feedrate can be adjusted here.



See Configuring FlashCut for more details.

10.2 Spindle Settings

If a spindle Fab Head has been configured, its settings can be adjusted here.



1. Clearance Height

   Height of the spindle during rapid moves.

2. Plunge Start Height

   Height of the spindle at the start of a download cut.

3. Peck Retract Distance

   Distance the spindle will move away from the workpiece during peck operations.

10.3 Knife Settings

If a knife Fab Head has been configured, its settings can be adjusted here.

1. **Feedrate**

   Rate when cutting.

2. **Plunge Rate/Retract Rate**

   Rate during negative lift axis movements.

3. **Lift Height**

   Height of cutting head above workpiece during cuts.

4. **Safe Height**

   Height of cutter during rapid moves.

5. **Maximum Rotation While Engaged**

   Degrees of rotation permitted.

6. **Corner Looping**

   When selected, the cutting head will describe a loop at the corners of a part for cleaner cuts.

   - Looping – Select Rounded or Triangular loop shape
   - Extension Distance – Distance of loop beyond corner
   - Maximum Angle to Add Loop – Largest allowed angle

7. **Corner Undercut Tolerance**

   Set value.

8. **Minimum Cutting Radius**

Set value.

9. **Overcut Depth**

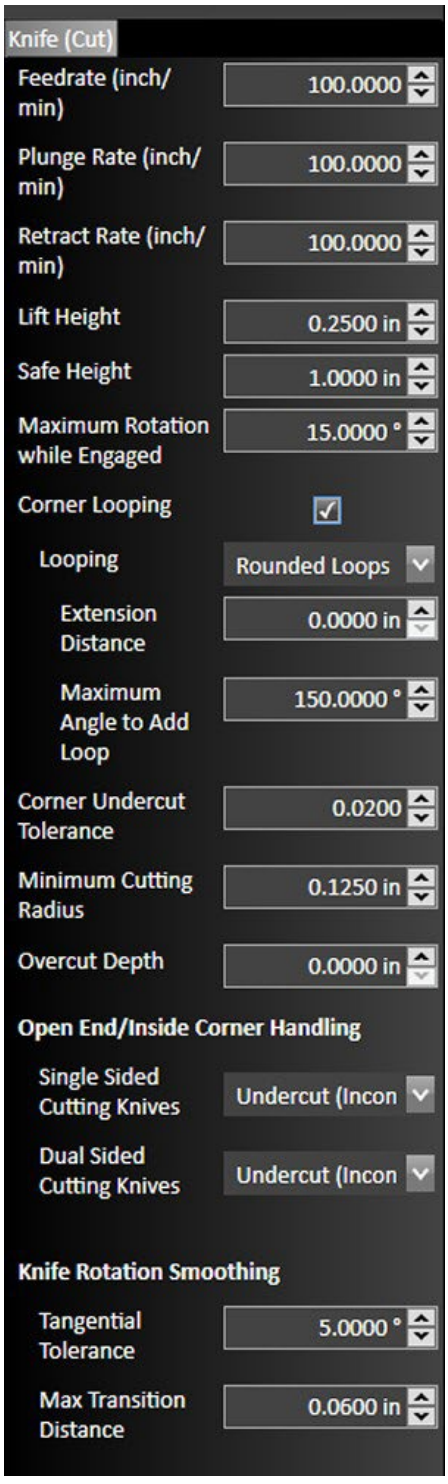
   Distance of cut through workpeice.

10. **Open End/Inside Corner Handling**

    - Single Sided Cutting Knives
      - Undercut (Incomplete Cut)
      - Cut to Vertex (Allow Gouging)
      - Stop Short and Plunge
    - Dual Sided Cutting Knives
      - Undercut (Incomplete Cut)
      - Cut to Vertex (Allow Gouging)

11. **Knife Rotation Smoothing**

    - Tangential Tolerance
    - Max Transition Distance



10.4 Grid Nesting

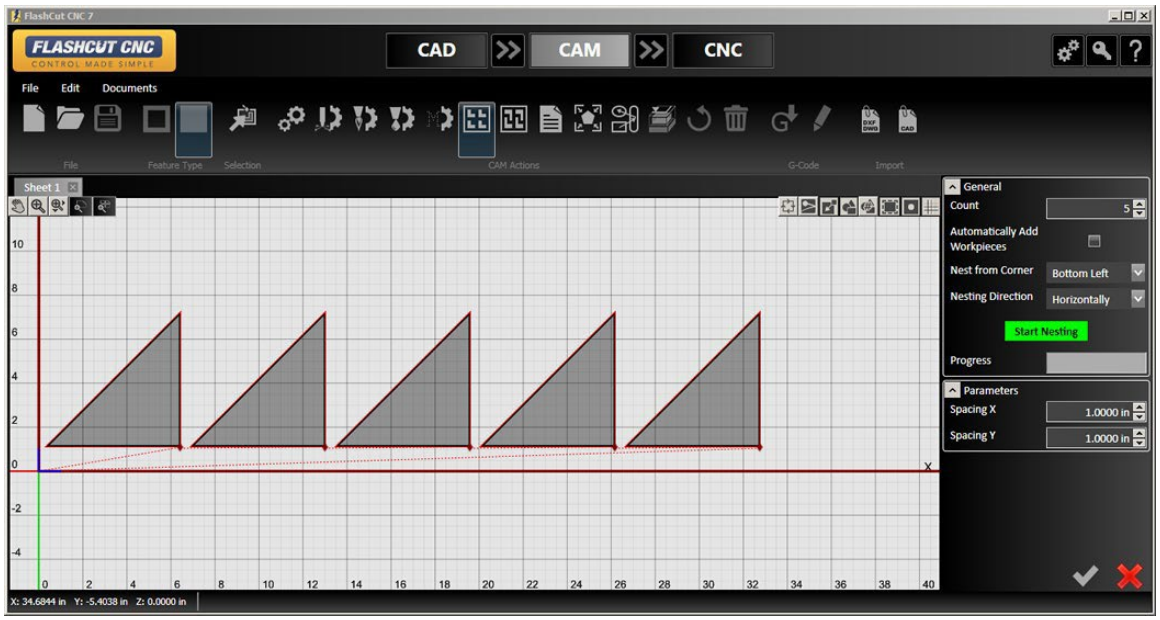
The grid nesting function duplicates a part in a grid pattern. In the parameter window, type in the count of parts that you want to cut along with the X and Y spacing of parts in the grid. The number of rows and columns is determined by the sheet size.

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 as illustrated in the screen below where five (5) triangles are grid nested. The parameters for grid nesting are described below.

1. **Count**

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

2. **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.

3. **Nest from corner**

   Selects the starting point of the nesting operation.

4. **Nesting direction**

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

5. **Start Nesting**

   Starts the nesting operation.

6. **Progress**

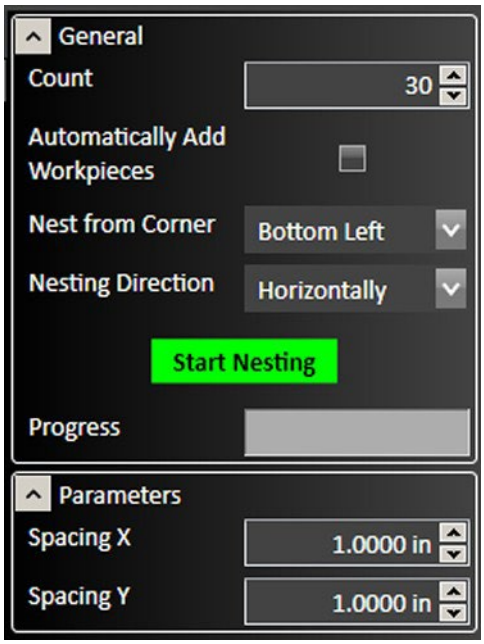
   Shows the progress of the nesting operation. Complex nesting operations can take significantly longer.

7. **Spacing X**

   Sets the horizontal spacing between parts.

8. **Spacing Y**

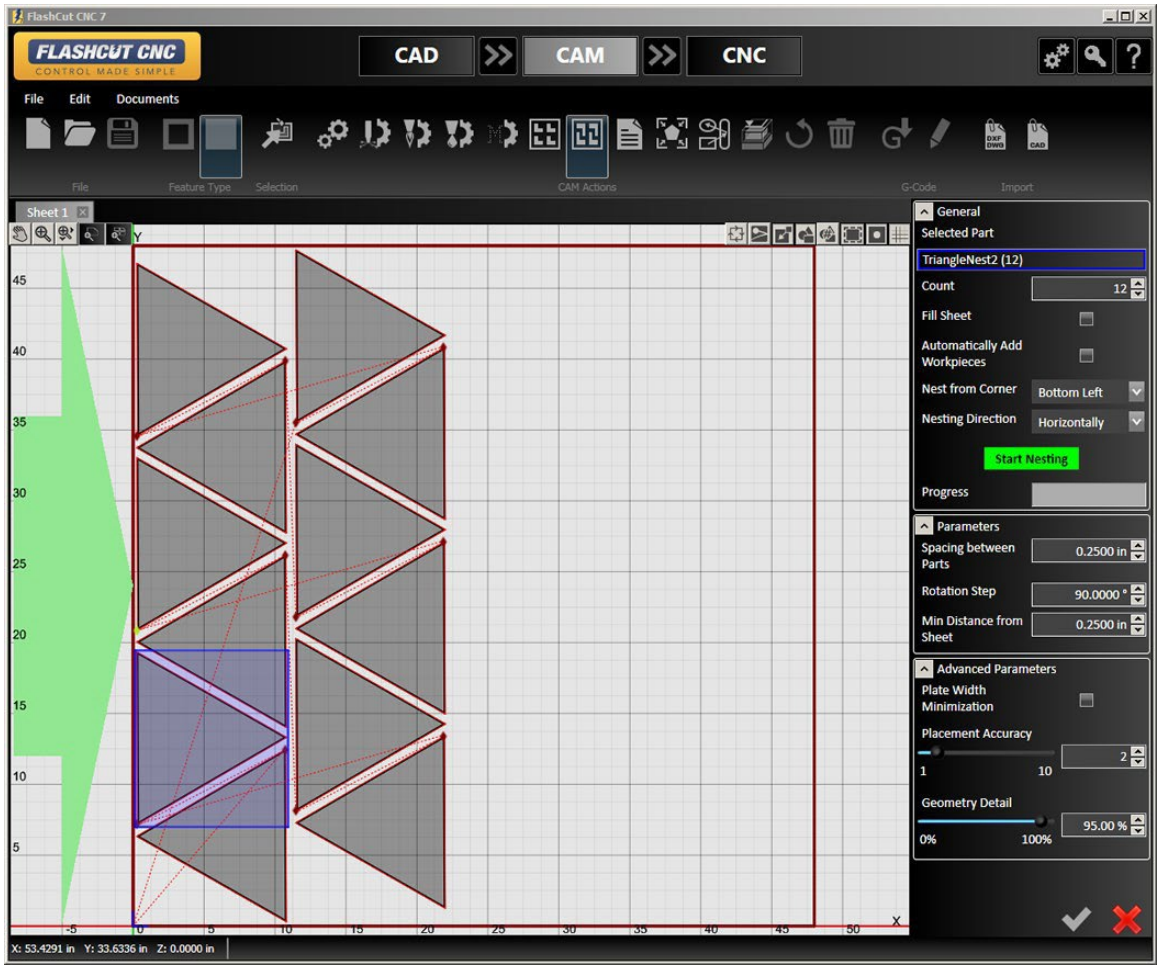
   Sets the vertical spacing between parts.



10.5 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 (12) 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.



1. **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.

2. **Count**

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

3. **Fill Sheet**

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

4. **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.

5. **Nest from Corner**

   Selects the starting point of the nesting operation.

6. **Nesting Direction**

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

7. **Start Nesting**

   Starts the nesting operation.

8. **Progress**

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

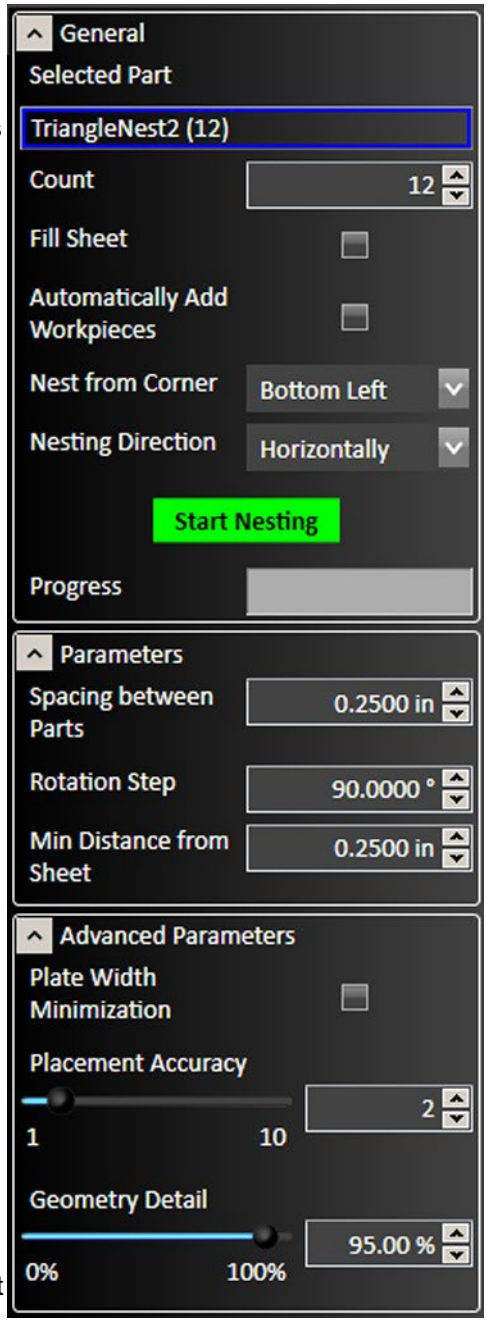
9. **Spacing between Parts**

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

10. **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.

11. **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.

12. **Plate Width Minimization**

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

13. **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.

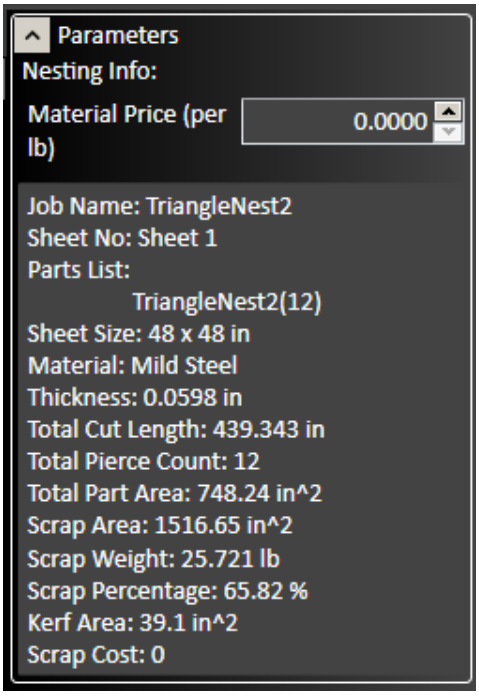
14. **Geometry Detail**

    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%.

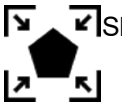
10.6 Nesting Information

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.



10.7 Fit Workpiece to the Parts

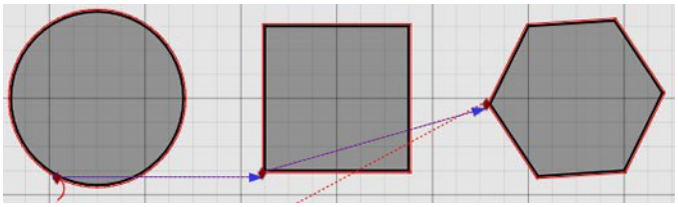


Shrinks the workpiece to fit the area defined by the parts.

10.8 Sequence Tool

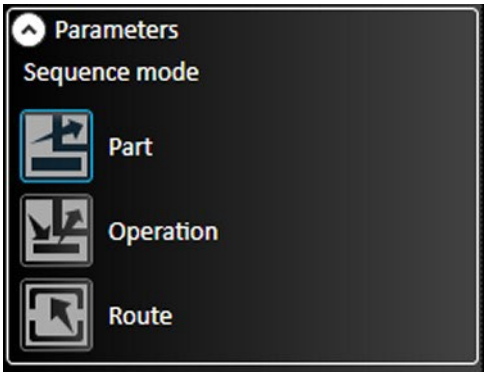
When selected, the sequence tool will display the order in which parts will be cut or operations executed. The example below shows a trio of parts being cut from left to right.

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.



1. **Part Sequence Mode**

   In this mode, FlashCut will execute all operations to complete one part before proceeding to the next part.



2. **Operation Sequence Mode**

   In this mode, individual operations can be re-ordered. For example, all marking operations can be executed before cutting operations.

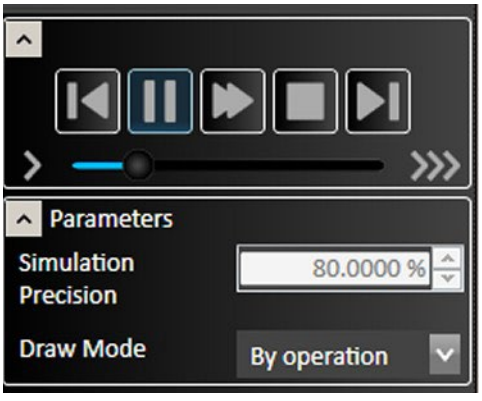
3. **Route Sequence Mode**

   In this mode, the moves between operations can be highlighted and the operations re-ordered. When the top-level feature of a drawing is a cutout, operation sequence mode is the only mode available.

10.9 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.

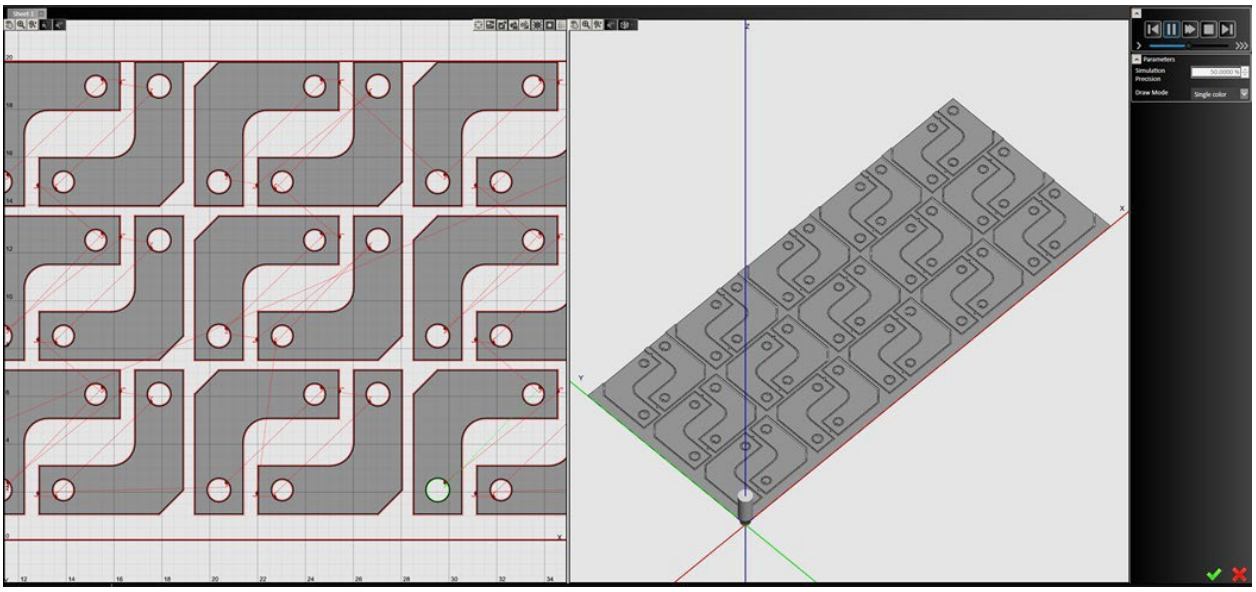
The slider with the arrows controls the speed at which the simulation plays.

1. Simulation Precision

Range: 1-100%.

2. Draw Mode

   Single color or by operation.



10.10 CAM Reset

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

10.11 Delete

Deletes the selected tool path.

10.12 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. These are the tools that are available on the G-Code portion of the ribbon.

- Create G-Code file
- Open Editor

10.13 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.

10.14 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.

10.15 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.

10.16 DXF/DWG

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

10.17 CADCAM

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

10.18 Pan and Zoom Tools

The pan and zoom tools are on 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.

1.  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

2.  Zoom Workpiece Extents

   Click this button to zoom the drawing to fill the workspace with the workpiece.

3.  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.

4.  Zoom Toolpath Extents

   Click this button to zoom the drawing to fill the workspace with the toolpaths.

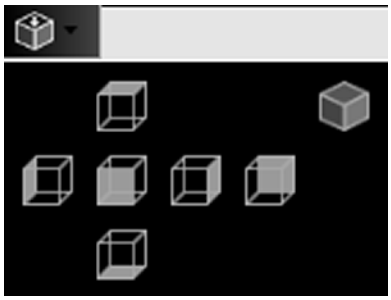
5.  Zoom Window

   Click and drag a box in which to zoom in. This tool allows you to focus in on a specific section of the diagram.

6.  Orientation Tool

   The tool palette for 3D part drawings will also display the orientation tool. The drawing workspace can be set to display any face of a three-dimensional object or an isometric view (default).

   Position the cursor over the orientation tool icon to display and select either a specific face of an orthographic view (front, top, left, etc.) or an isometric view.



10.19 Display Options Tools

The display options are shown at the top of the drawing workspace and allow you to decide what appears in the drawing

window. Click the buttons to toggle each option on and off. You may toggle any number of options at one time.

10.20 Cutting Directions

When toggled, the direction of motion of the cutting tool is shown in the drawing as a series of small arrows.

10.21 Kerf

The kerf indicates the width of the cut created by the plasma torch or other cutting tool. When toggled, a blue highlight will appear over the cut outline which indicates exactly how the feature will be cut. You can modify the kerf width in Plasma Settings.

10.22 Material

When toggled, a red box is displayed in the drawing to indicate the actual sheet dimensions. Double clicking this box enables you to edit its dimensions in the parameter window. You can also click on the perimeter of the sheet to bring up the parameter window.

10.23 Parts

When toggled, each feature to be cut it outlined in black on the CAM drawing.

10.24 Part Numbers

When toggled, the name and number of each part appears next to it in the drawing window.

10.25 Rapid Moves

When toggled, rapid moves of the cutting tool are displayed on the drawing as dotted red lines. In order for rapid moves to be displayed, Tool Paths must also be toggled.

10.26 Tool Paths

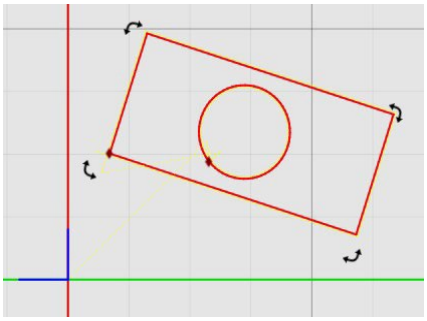
When toggled, the path of the cutting tool is outlined in red on the CAM drawing.

10.27 Drawing Workspace

This section discusses the following topics:

- Feature manipulation
- Lead in/out settings
- Program zero

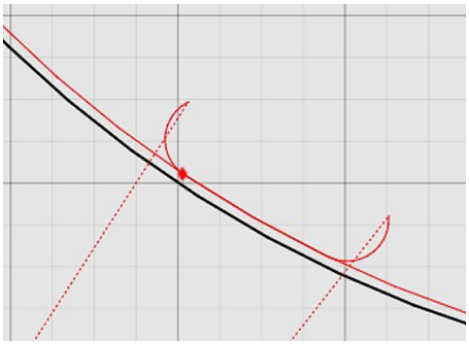
10.28 Feature Manipulation



You can maneuver parts in the drawing window into a desired rotation or position. Move the cursor over the part so that is highlights green and select it so it turns red.

From here, you may click and drag the part to a specific location. You can also rotate parts by clicking and dragging one of the arrows in the corners. Moving a feature also adjusts automatically generated features, including lead in, lead out lines, and rapid move paths.

10.29 Lead in/out Settings

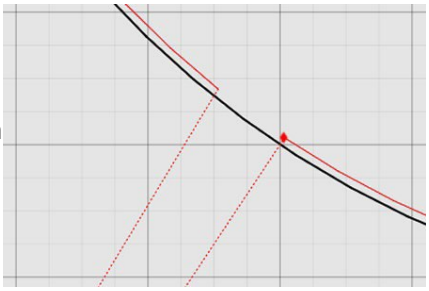


Lead in and lead out lines are used to provide a way for the torch to ease into a shape. Usually, they are oriented in the same direction as the tool path in order to ensure a clean cut of the final piece. The lengths of these lines are initially calculated from the thickness of the workpiece.

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.

10.30 Editing Settings

To adjust lead in and lead out settings, select the red diamond icon on the desired feature. This brings up the parameters window, select the red diamond icon on the desired feature. This brings up the parameters window.

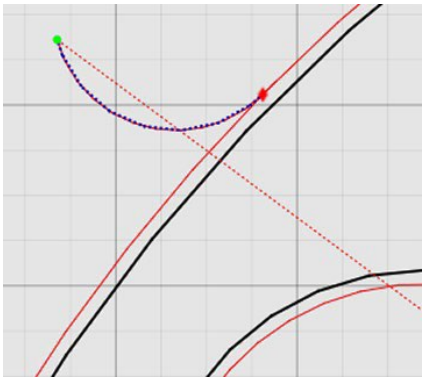


If you want to change the location of the lead in and lead out, then you can simply drag the red diamond along the tool path to the desired location.

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.

10.31 Manual Editing

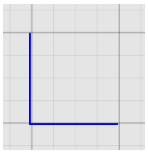


FlashCut CAM permits manual adjustment of individual lead ins and lead outs in the tool path.

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).

10.32 Program Zero



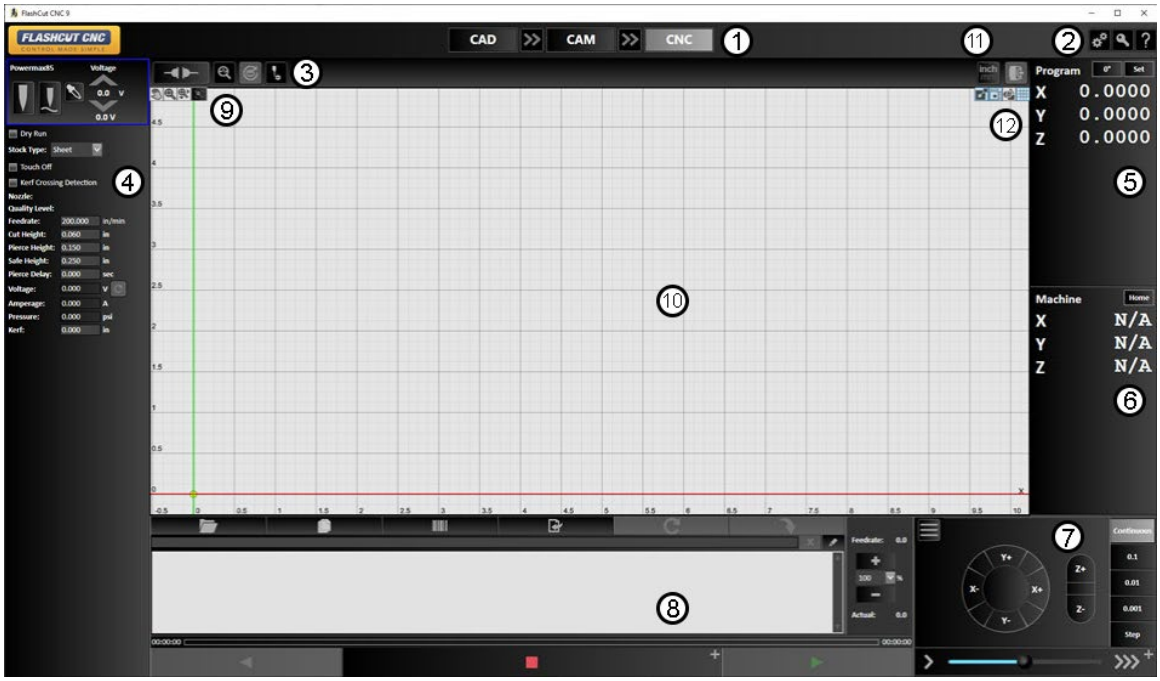
The blue set of axes in the drawing window represents the origin of the program coordinates. You may move these axes around by clicking and dragging the blue lines or by entering coordinates manually in the parameter window.

NOTES:

11.0 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 (2) 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.



1. Tabs
2. Configuration, License, and Help buttons
3. System Status, Connect, Reset Motor Drivers and Tool Rack Positions
4. Fab head settings
5. Program coordinates panel
6. Machine Coordinate Panel
7. Jog and point control panel
8. G-Code Window
9. Parameters area
10. Drawing workspace
11. Toggle Display Units
    Toggle manual control
12. Material
    Machine Envelope
    Part Numbers
    Show/Hide Grid

11.1 System Status

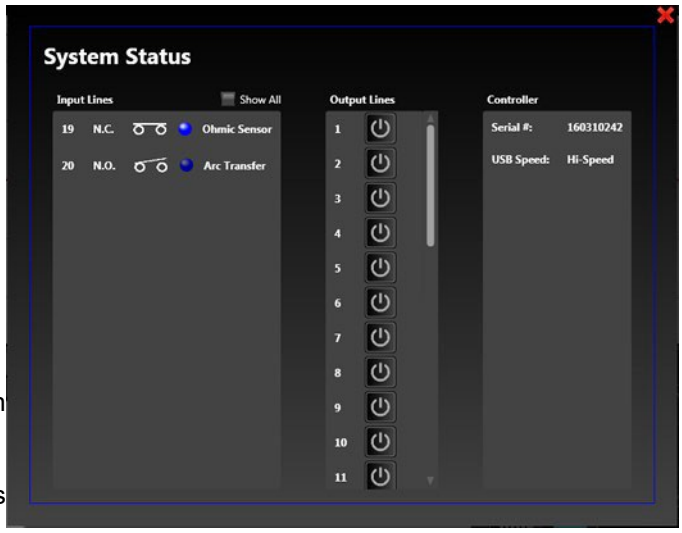
Click the magnifying glass icon to view the live status of the input and output lines.

11.1.1 Input Lines

If a normally closed (NC) input line a 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 when 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.



11.1.2 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.

11.1.3 Controller

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

11.1.4 Connect

Clicking this icon connects or disconnects the signal generator. When connected, the icon is illuminated and the two (2) 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 (see Installing the USB Driver for instructions).

Click the Connect icon. FlashCut begins communication with signal generator. If there is a problem with the connection, a dialog box appears.

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.

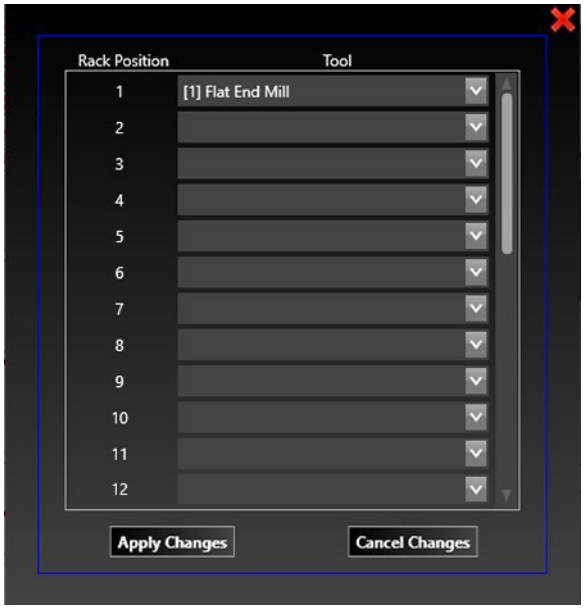
11.1.5 Reset Motor Drivers

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

11.1.6 Tool Rack Positions

Click to view and change tool rack positions.

Each rack position displays a dropdown field with available tools. See Tool Library for more information.



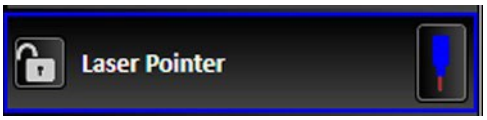
11.2 Fab Head Settings

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.

11.3 Laser Pointer Control

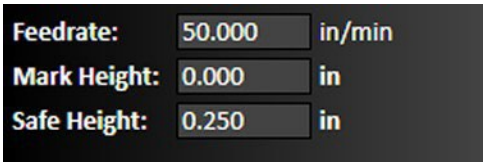
If a laser pointer is configured, it can be turned on and off by clicking the icon. The pointer can also be locked in the on position.



11.4 Marker Control

If a marker is configured, it can be turned on and off by clicking the icon.

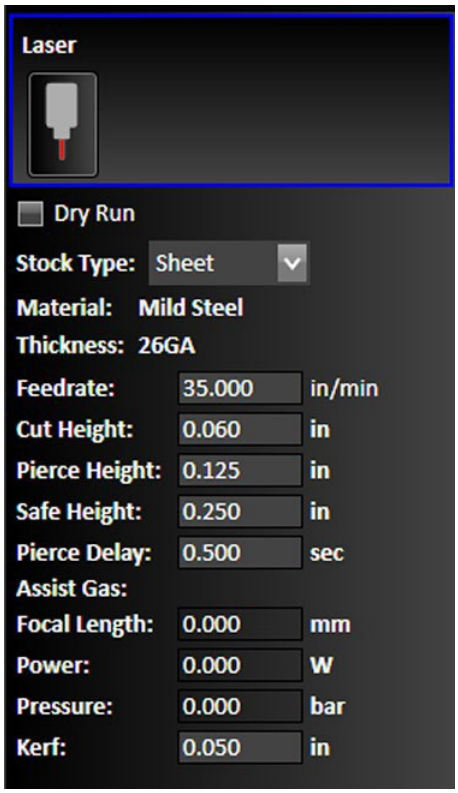
- **Feedrate** – in system units/min.
- **Mark Height** – Specifies the distance between the marker tip and the stock material throughout a marking operation.
- **Safe Height** – Specifies the height to which the Fab Head is set during rapid moves.



11.5 Laser Cutter 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.

- **Laser On/Off** – Click to turn the laser on or off. Before turning on the laser, FlashCut CNC will display a warning dialog asking if it is safe to proceed. Controls for other options (e.g. Marking) will appear here if configured.
- **Dry Run** – When selected, the machine will execute the operations, but the laser will not activate and cuts will not be made.

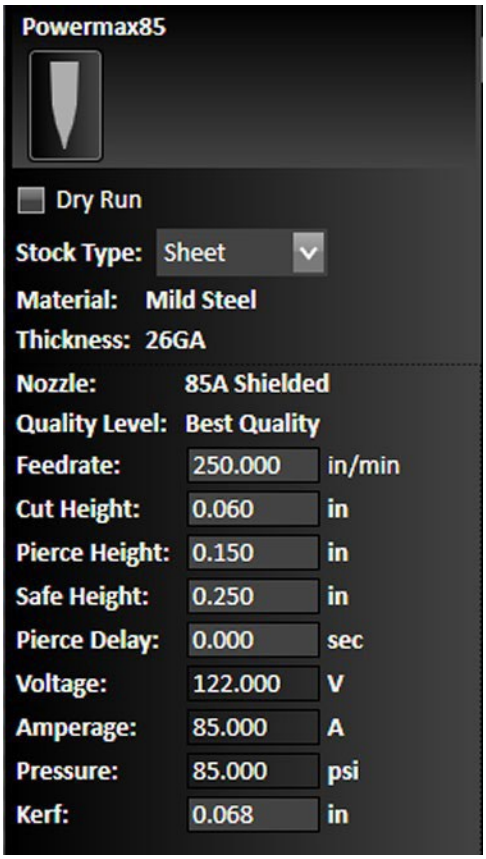


- **Stock Type, Material, and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Cut Height** – Specifies the distance between the torch tip and the stock material throughout a cutting move.
- **Pierce Height** – Specifies the distance between the torch tip and the stock material at the time the laser is activated.
- **Safe Height** – Specifies the height to which the laser is set during rapid moves.
- **Assist Gas**
- **Focal Length**
- **Power**
- **Pressure** – Pressure (psi) of the Fab Head's assist gas jet.
- **Kerf** – Specifies the width of the kerf used by the cutter.

11.6 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.

- **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.
- **Dry Run** – When selected, the machine will execute the operations, but the torch will not be lit and cuts will not be made.
- **Stock Type, Material, and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Nozzle** – Displays information about the plasma torch nozzle.
- **Quality Level** – Specifies whether the cut will be optimized for speed or quality.



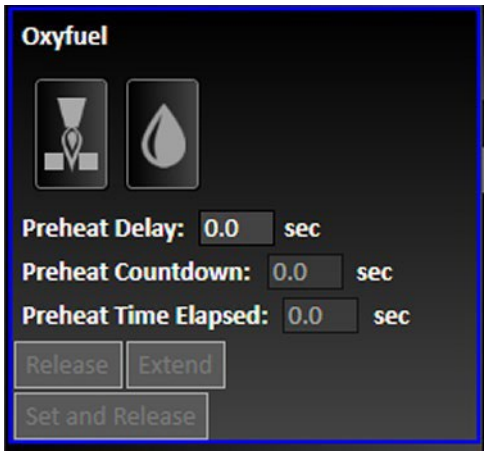
- **Feedrate** – in system units/min.
- **Cut Height** – Specifies the distance between the torch tip and the stock material throughout a cutting move.
- **Pierce Height** – Specifies the distance between the torch tip and the stock material at the time when the torch is commanded to fire.
- **Safe Height** – Specifies the height to which the torch is set during rapid moves.
- **Pierce Delay** – Specifies the delay between the command to fire the torch and the start of the next move.
- **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.

11.7 Oxyfuel Settings

Parameters are populated from Settings set in the CAM window. If an edible 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.

- **Control Buttons** – For cutting and water.
- **Preheat Delay** – Specifies the time to preheat the material before cutting.
- **Preheat Countdown**
- **Preheat Time Elapsed**
- **Stock Type, Material, and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Pierce Delay** – Specifies the time to delay before completing a pierce operation.

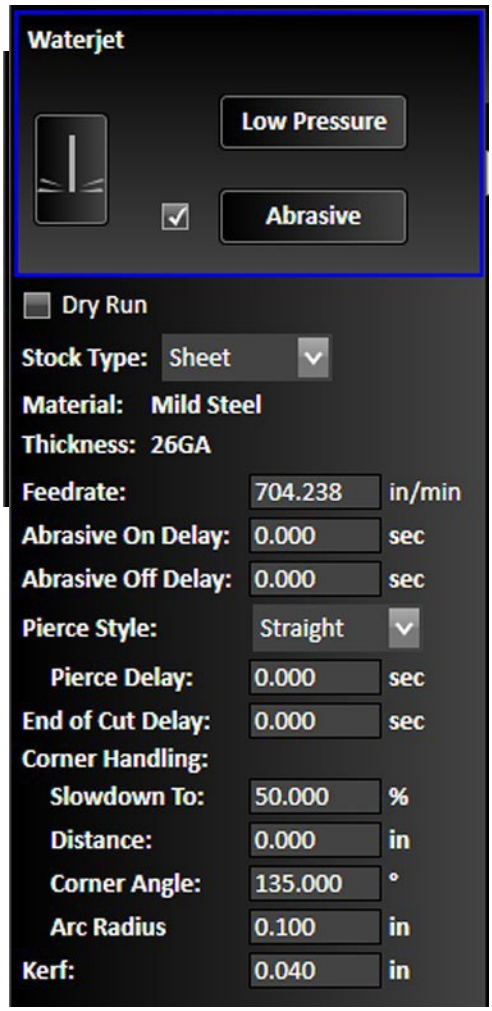
- **Purge Delay** – Specifies the time to delay before completing a purge operation.
- **Use Water** – Toggles this feature, if enabled.
- **Kerf** – Specifies the width of the kerf used by the cutter.
- **Creep Rate**
- **Creep Time**



11.8 Waterjet 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.

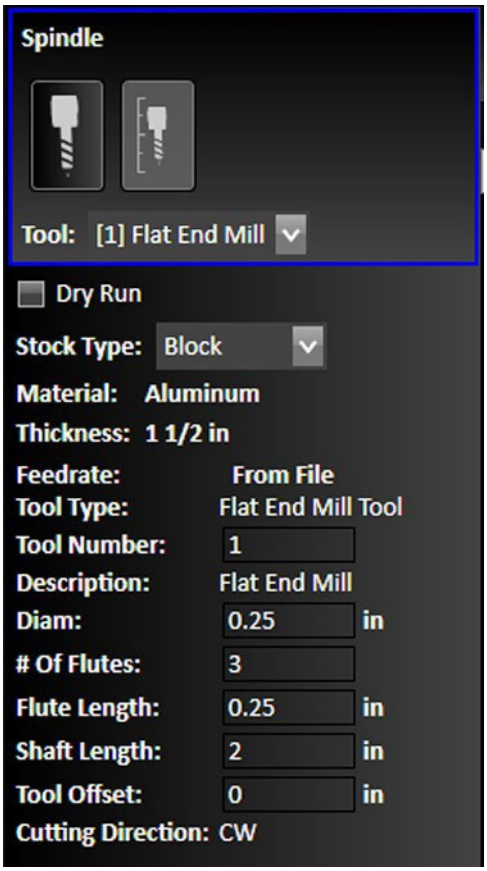
- **Control Buttons** – Click to turn the jet on or off. If configured, buttons to select Low Pressure and Abrasive will also be visible.
- **Dry Run** – When selected, the machine will execute the operations, but the jet will not be activated, and cuts will not be made.
- **Stock Type, Material and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Feedrate** – in system units/min.
- **Abrasive On Delay** – Specifies the time in seconds before the flow of abrasive stars after the cutter is activated.
- **Pierce Style** – Straight is shown. Additional configuration options are visible for specific types (e.g., Radius for Circular).
- **Pierce Delay** – Specifies the time to delay before completing a pierce operation.



- **End of Cut Delay** – Specifies the delay at the end of the cut. The purpose of the delay is to make sure the cut is complete.
- **Corner Handling** – Settings to enable cleaner corner cuts: Slowdown percentage, Corner Angle, and Arc Radius.
- **Kerf** – Specifies the width of the kerf used by the jet.

11.9 Spindle Settings

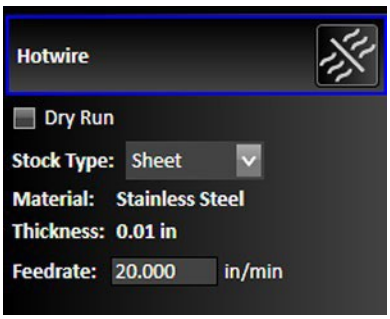
Parameters are populated from Settings set in the CAM window. If an edible parameter is changed here, the settings in CAM will not be affected. Some controls will not be displayed unless they are enabled in the fabrication head configuration panel.



- **Control Buttons** – Turn on the spindle. Before turning on the spindle, FlashCut CNC will display a waring dialog asking if it is safe to proceed.
- **Tool** – Select tool. Must be defined in tooling file.
- **Dry Run** – When selected, the machine will execute the operations, but the spindle will not be activated, and cuts will not be made.
- **Stock Type, Material, and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Tool Type, Tool Number, and other parameters** – Defined in the tooling configuration file. See Tooling Configuration Tab and Fabrication Heads in Configuring FlashCut.

11.10 Hotwire Settings

The HotWire Fab Head can only be used directly in FlashCut CNC. See Configuring FlashCut.



- **Control Buttons** – Turn on the HotWire. Before turning on the Fab Head, FlashCut CNCN will display a warning dialog asking if it safe to proceed.
- **Dry Run** – When selected, the machine will execute the operations, but the spindle will not be activated, and cuts will not be made.
- **Stock Type, Material, and Thickness** – These properties are visible here, but are configured in the CAM window under Project Settings.
- **Feedrate** – Set machine feedrate here.

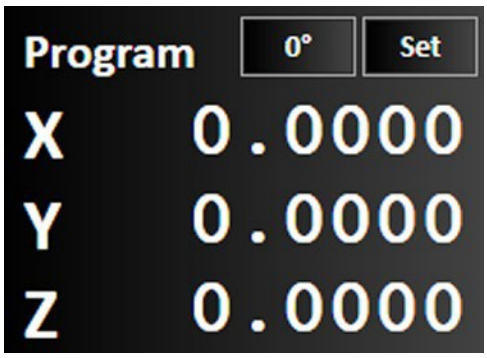
11.11 Tangential Knife Settings

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.

- **Control Button –** Turn on the spindle. Before turning on the spindle, FlashCut CNC will display a warning dialog asking if it is safe to proceed.
- **Tool –** Select tool. Must be defined in tooling file.
- **Dry Run –** When selected, the machine will execute the operations, but the knife will not be activated, and cuts will not be made.
- **Stock Type, Material, and Thickness –** These properties are visible here, but are configured in the CAM window under Project Settings.
- **Tool Type, Tool Number, and other parameters –** Defined in the tooling configuration file. See Tooling Configuration Tab and Fabrication Heads in Configuring FlashCut

11.12 Program Coordinates Panel

The program coordinates refer to the exact position of the tool width 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 panel has two (2) dropdown menus:

- **Coordinate Rotation (0º)** sets the rotation of the drawing window.

- **Set** sets the zero point

The **Coordinate Rotation** dropdown menu **(0º)** sets the rotation of the drawing window. When the coordinates of the drawing window are rotated, all operations will be adjusted to the degree of rotation set. The degree of rotation may be set either by defining two (2) corners in the drawing workspace, or by inputting a numerical value.

- **2-point alignment** shows the corners that have been defined in the drawing workspace. These points are defined by manually moving the tool to a coordinate value (for example, by using the Jog and point control panel) and then selecting an item from the dropdown menu. Options include:
    - Lower Left and Lower Right
    - Lower Left and Upper Left
    - Lower Rights and Upper Right
    - Upper Left and Upper Right
- **Enter Rotation Angle** allows manual entry of a numerical value in thousands of degrees. The angle may be positive or negative.
- **Clear Rotation** resets the angle of rotation to zero degrees.

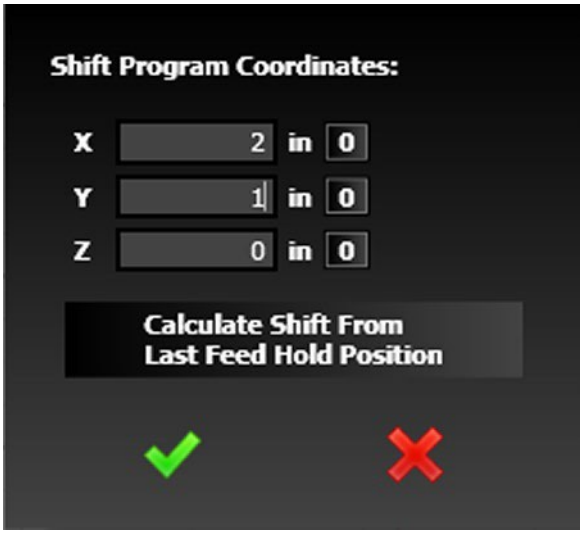
An example with step-by-step instructions appears below:

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.
- **Level Rotating Stock**
- **Select Offset**
- **Define Offset**

NOTE

The operation of the NEW and Shift commands has been simplified in version 7.04 and higher. Unless a value is specified, the value for X, Y, or Z will be 0. The value for any axis can be set to 0 by clicking the button to the right. The Shift window is shown below.



11.13 Coordinate Rotation Example

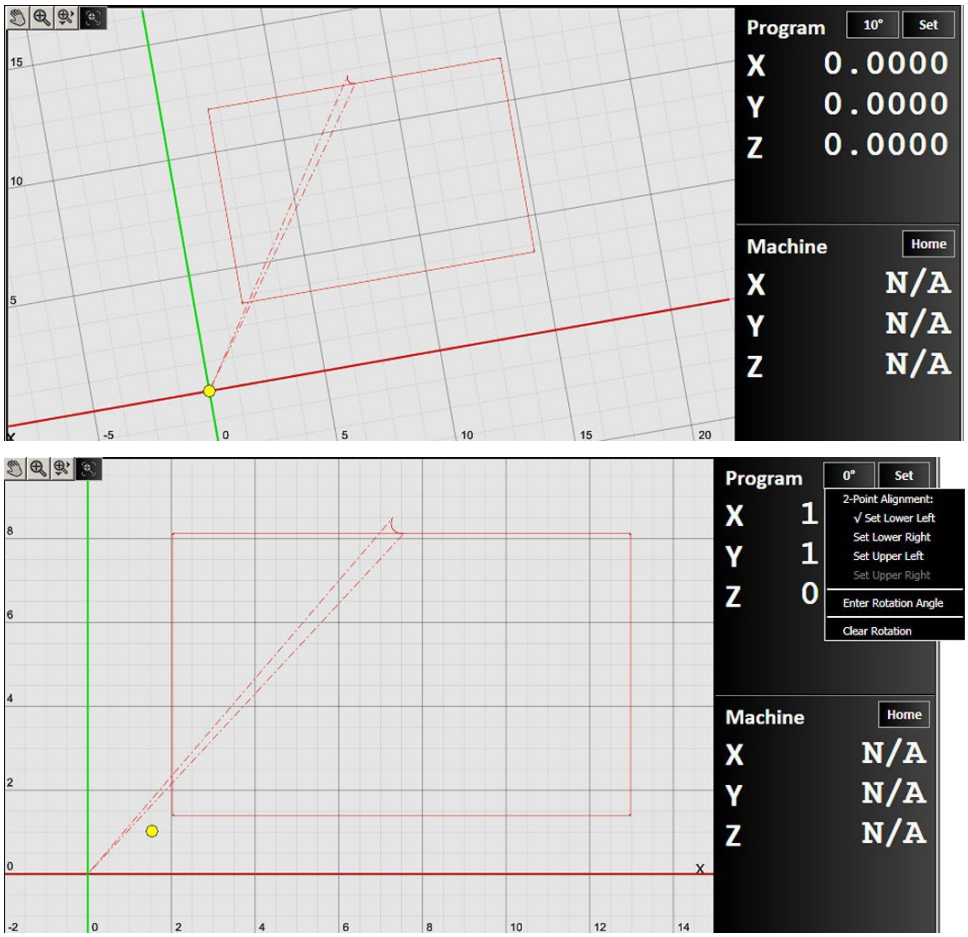
Coordinates rotation can be used to accommodate a workpiece that is too heavy or inconvenient to move on the machine. In effect, the program axes can be adjusted to confirm the dimensions and orientation of the workpiece.

11.14 Enter Rotation Angle

If the desired angle of rotation is known that angle can simply be entered.

1. Select **Enter Rotation Angle** from the dropdown menu.
2. Enter the angle. The value may be positive or negative.

The angle of rotation will be displayed in the Program coordinates panel and the drawing workspace will be rotated.



11.15 Use 2-Point Alignment

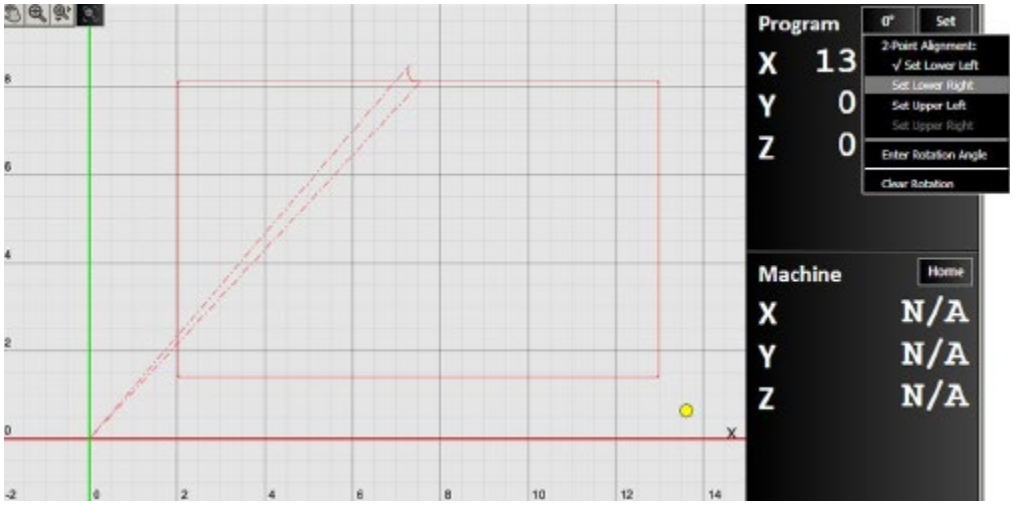
When using 2-point alignment, the position of the tool is used define two (2) points on the workpiece that define the angle of rotation. The angle of rotation is calculated by comparing the relative positions of two (2) points selected.

1. Jog the tool position to the lower right corner of the workpiece.
2. Select **Set Lower Left** from the dropdown menu.
3. Jog the tool position to the lower right corner of the workpiece.
4. Select **Set Lower Right** from the dropdown menu. Note that invalid options are greyed out in the menu. In the figure below, the menu item is chosen, but not yet selected.

5.

6. Use the **Set** dropdown menu to return the tool position to program zero.

7. Select **Set Lower Right** from the dropdown menu. Note that invalid options are greyed out in the menu. In the figure below, the menu item is chosen, but not yet selected.

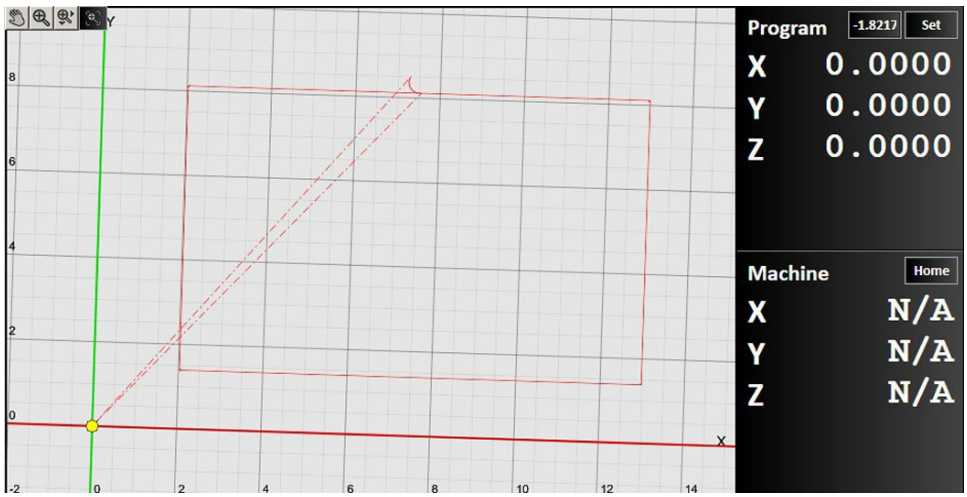


8. Use the **Set** dropdown menu to return the tool position to program zero.

Again, the value will be displayed in the Program coordinates panel and the drawing workspace will be rotated.

11.16 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 thee (3) 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.

When multiple motors are used to drive an axis, FlashCut simultaneously homes each motor independently. For example, if two (2) motors are driving a single axis, FlashCut starts the homing process by moving both motors towards the homing limit switches. Once one motor reaches its limit switch, it stops and waits until the other motor reaches its limit switch. Once that occurs, all motors retract simultaneously from the limit switches. This process corrects any skewing that may have occurred between the two (2) actuators.

11.17 Jog and Point Control Panel

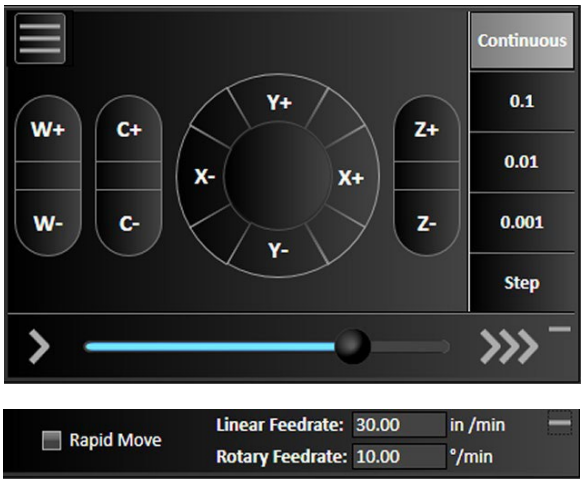
The jog and point control panel allows you to move your tool location in two (2) ways. It also allows you to select the rip cut feature, when available.

This icon in the top left corner of the panel brings up a menu that allows you to select which panel to display:

- Jog
- Move to point
- Rip Cut

11.18 Jog

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



11.19 Axis Jog Buttons

Pressing and holding an axis jog button (X/Y/Z/W/C) 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.

11.20 Diagonal Jog Buttons

Pressing and holding a diagonal jog button (unlabeled) moves the machine tool evenly along two (2) 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.

11.21 Jog Mode

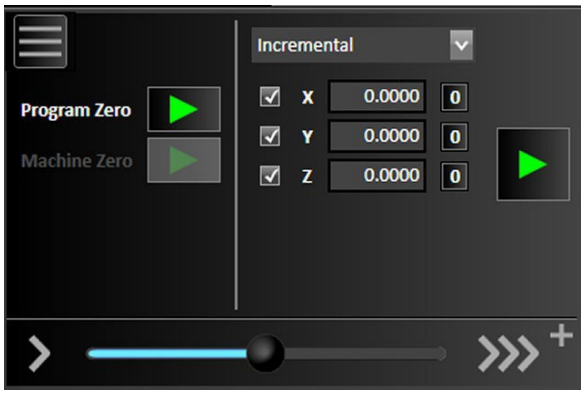
Determine the function of the jog buttons. **Continuous** jogging moves the tool at the assigned jog rate buttons are held down. **Discreet distances** moves the tool incrementally the specified distance. **Step** jogging advances the motor exactly one more step.

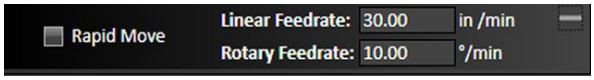
11.22 Jog Rate

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.

11.23 Move to Point

Move-to-point allows you to move (jog) your tool to a specified exact point. You may specify the feedrate 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.

You may also use the controls on the right side of the jog-to-point panel to modify the destination based on your selection with the dropdown menu. The choice you make with the dropdown menu specifies which point the move will be made with respect to. You may choose from:

- **Program** – When you choose Program, the tool can be moved with respect to the program zero point shown on the **Program coordinates panel**.
- **Machine** – When you choose Machine, the tool can be moved with respect to the program zero point shown on the **Machine Coordinates Panel**.

- **Incremental** – When you choose Incremental, the tool can be moved with respect to its current location.

Check the box next to the axis you want the move to be made in. You may make the move in one (1), two (2), or all three (3) axes. Enter a positive or negative numeric value in the appropriate field to instruct FlashCut to move that distance from the specified location. You may click the 0-button next to a field to automatically populate that field with 0.0000. When you've specified the distances, you want the tool to move, click the green arrow control to start the desired motion.

11.24 Move Rate

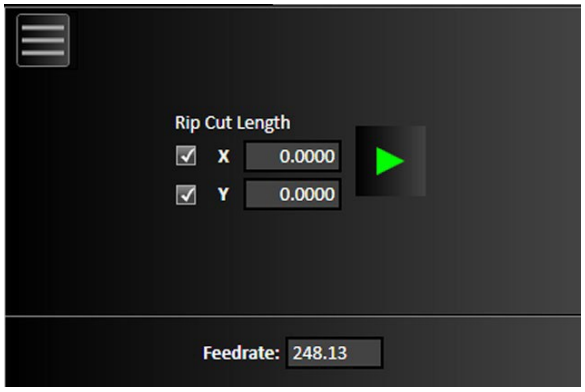
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

11.25 Rip Cut

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 feedrate.

Press the green arrow to make the cut.



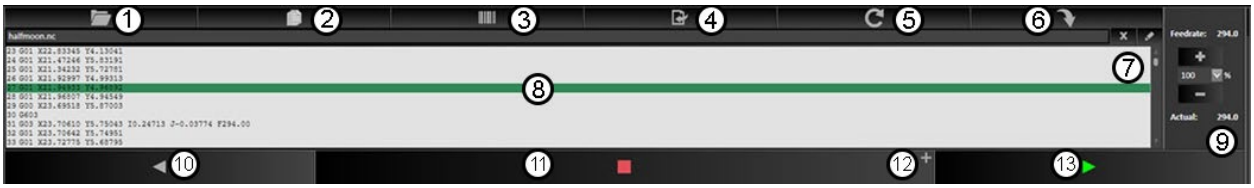
11.26 G-Code Window

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 and FlashCut CAM. Clicking the scissors button in either of these modes generates 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
2. Load all G-Code files in a folder
3. Barcode
4. Run CAD Import Wizard
5. Reset G-Code
6. Jump to Line
7. Closes G-Code file
   Edit G-Code
8. G-Code workspace
9. Feedrate override controls
10. Run G-Code in reverse
11. Feed hold
12. Toggle G-Code run mode
13. Run G-Code

11.27 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.

11.28 File Name

FlashCut CNC displays the file name of the currently displayed G-Code file here.

11.29 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.

11.30 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.

11.31 Reset G-Code

Click the Rest G-Code file button to reload the current G-Code file. This will not change the position of the tool.

11.32 Jump to Line

This button allows the user to jump or skip ahead to a specific line in the program.

11.33 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.

11.34 Edit G-Code

Click the Edit G-Code button to edit the currently displayed G-Code file with the G-Code editor.

11.35 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.

11.36 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.

11.37 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.

11.38 Feed Hold

Click the Feed hold button to pause execution of the G-Code file. The machine tool stops, ramping down 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.

11.39 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.

11.40 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.

11.41 Pan and Zoom Tools

The pan and zoom tools are used for moving and magnifying the drawing in the drawing workspace. Click on a specific tool to change the cursor's function.

1.  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.

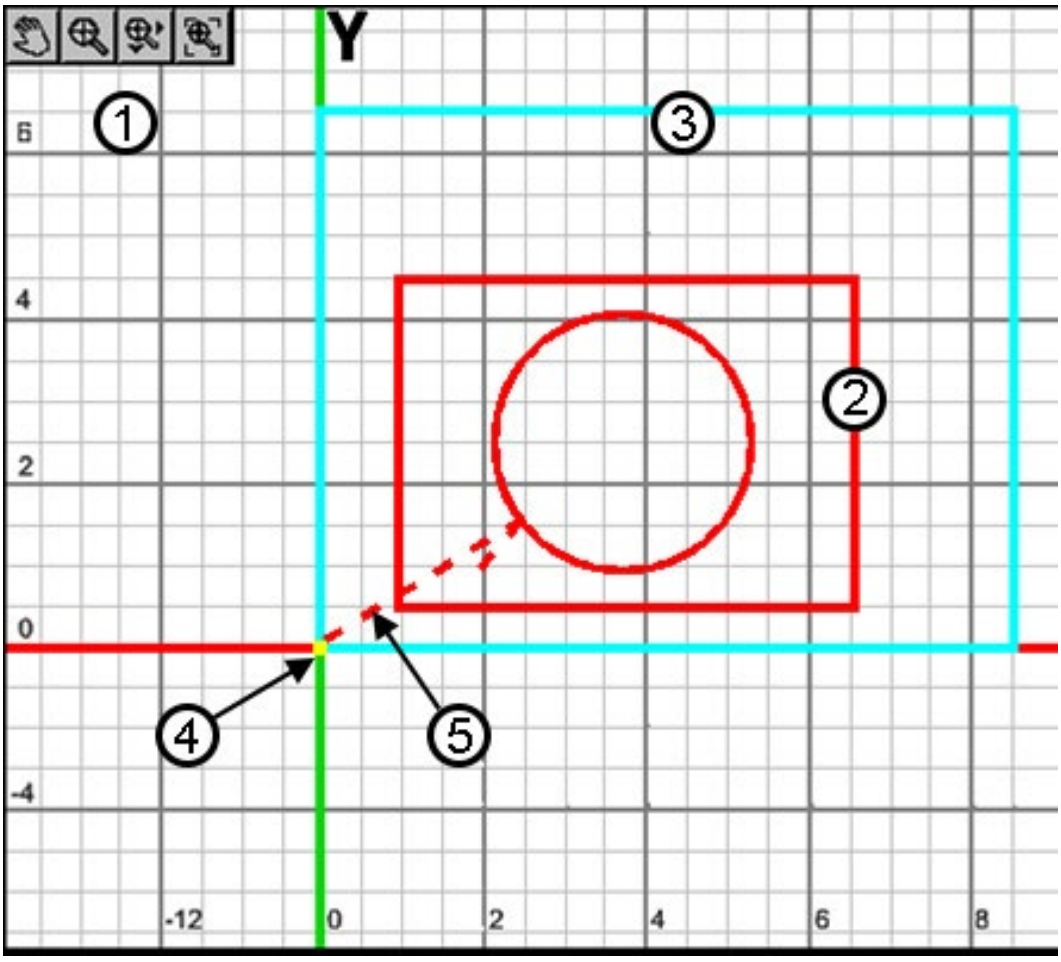
2.  Zoom – Click and drag anywhere to zoom in and out on the drawing. You can also use the scroll wheel to zoom in and out.

3.  Zoom Extent – Click and drag a box in which to zoom in. This tool allows the user to focus on a specific section of the diagram.
4.  Zoom to Fit – click this button to adjust zoom automatically to best fit the drawing.

11.42 Drawing workspace

The workspace displays different views of the tool path described by the current G-Code file.

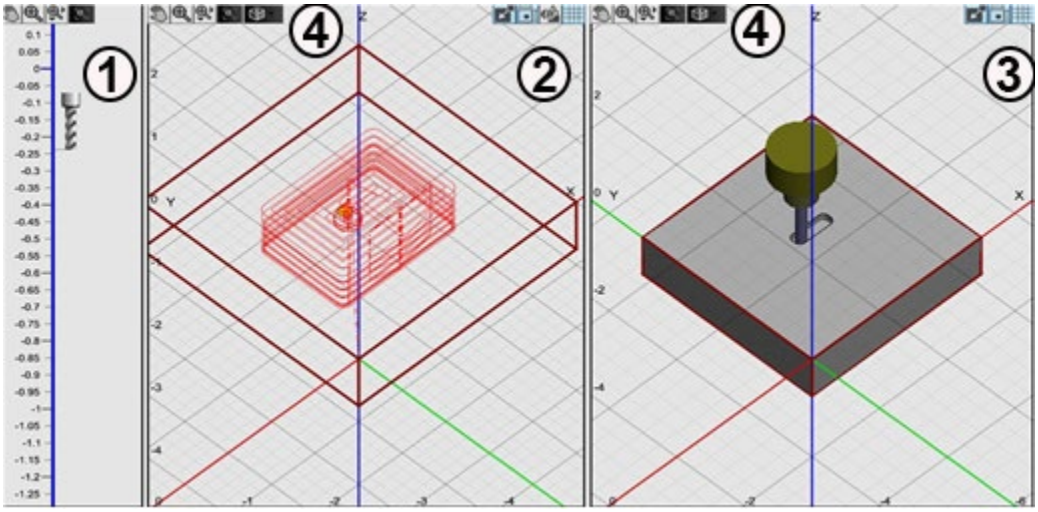
By default, FlashCut CNC displays a two-dimensional XY view. You may manipulate the two-dimensional view using the zoom and pan tools, as well as by using the Display options on the Preferences screen.



1. Pan and Zoom Tools
2. Solid red lines show the tool path
3. The maximum dimensions which a tool may not exceed are represented by a thick blue box. The box will appear after the machine has been homed. See Machine Envelope.

   To avoid improper machining, ensure that the entirety of the toolpath remains within this machine tool envelope. If the feature dimensions exceed this size, return to FalshCut CAD or FlashCut CAM in order to make revisions.
4. The current tool position is shown by a yellow dot.
5. Dotted red lines represented rapid moves

11.43 Multiple Views for 3D Operations



1 **Lift Axis View**

Displays the lift axis position of the Fab Head and tool during operations.

2 **Viewport**

Displays the toolpath and tool position.

3 **Simulation view**

Displays a simplified solid model of the workpiece and tool.

4 **Orientation Tool**

Both the viewport and the simulation view can be set to display any face of a three-dimensional object or an isometric view (default).

Position the cursor over the orientation tool icon to display and select either a specific face of an orthographic view (front, top, left, etc.), or an isometric view.

11.44 Toggle Display Units

Clicking this icon toggles the display units in the drawing window between inches and millimeters.

When inches are selected, inch will be highlighted in the icon. When millimeters are selected, mm will be highlighted.

11.45 Toggle Manual Control

Toggles back and forth between manual and servo mode. Allows manual adjustment while tracking the position of the tool.

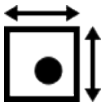
This icon is greyed out on machines without this capability.

11.46 Material

When selected, displays the edges of the workpiece in the drawing workspace.

11.47 Machine Envelope

When selected, displays the machine envelope. The machine must be configured and the controller must be connected for this option to be available.

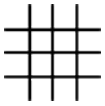


11.48 Part Numbers

When toggled, the name and number of each part appears next to it in the drawing window.

11.49 Show/Hide Grid

Toggles the grid in the active drawing workspace.



NOTES:

12.0 Configuring FlashCut

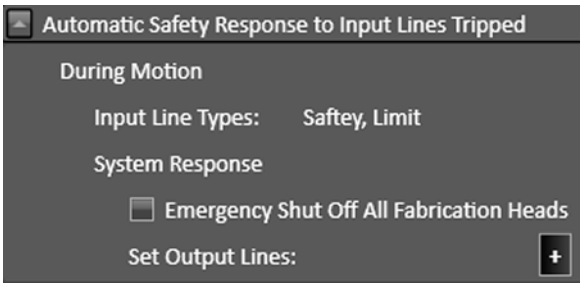
This section describes how to configure FlashCut for use with your machine tool. It is very important that the software and hardware are configured properly. Incorrect configuration may cause the machine tool to behave in a potentially dangerous manner. Please read through this section carefully to gain a thorough understanding of how FlashCut controls your machine.

NOTE

If an Administrator password has been set, you must enter it before you can make changes to the configuration. See Safety.

12.1 Automatic Safety Response

When selected, all fabrication will shut off when a safety or limit input line is tripped.

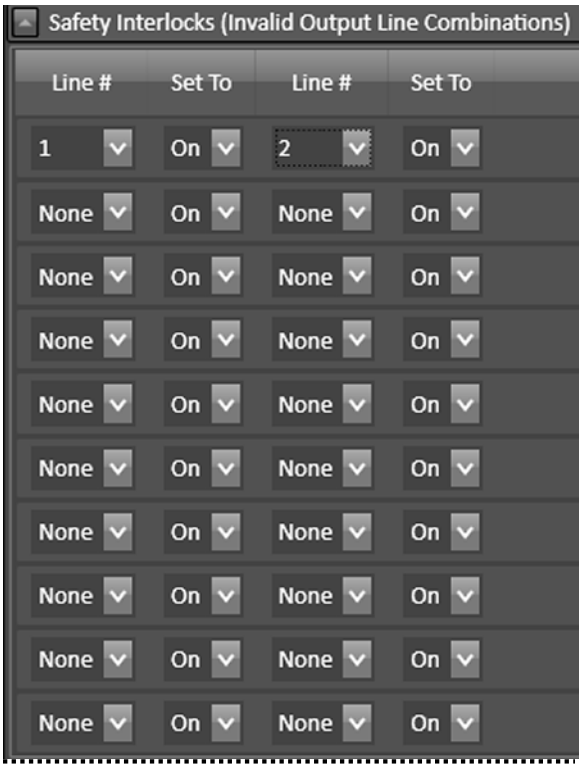


12.2 Safety Interlocks

The Safety Interlocks table lets you define incompatible combinations of output line on/off states. FlashCut will prevent the output lines from reaching a state that violates any of the safety interlocks you define.

To create an interlock, set the four (4) pulldown menus across a row of the table to define the incompatible states of two (2) output lines.

Example: The machine tool has an automatic tool changer that uses a spindle with a pneumatic tool chuck and the chuck should never be open when the spindle is turning. Assuming the spindle is on when output line 1 is on and the chuck is open when output line 2 is on, the four (4) pulldown menus on the first line of the table would be set as shown.



Output Line #: 1

Set To: On

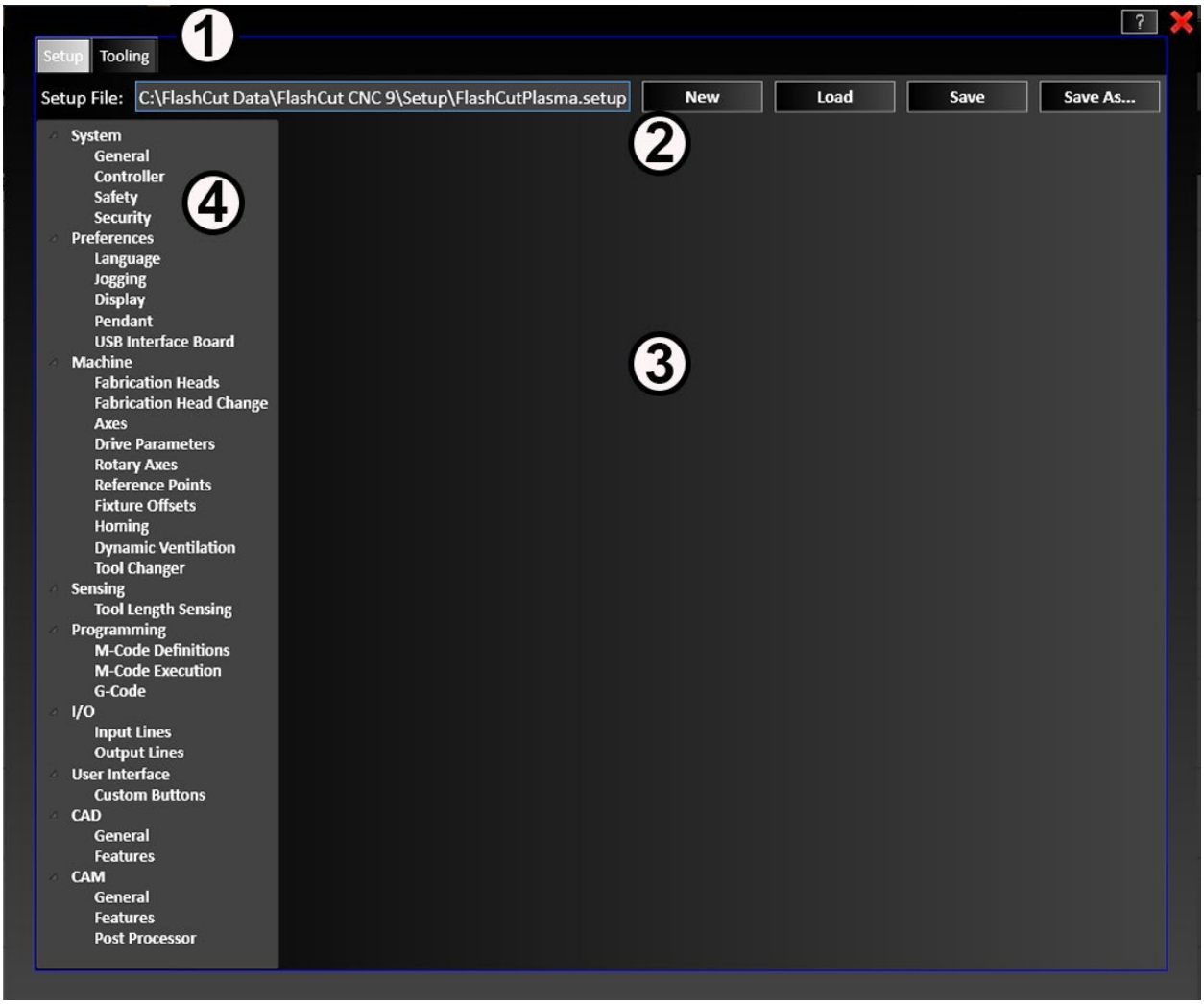
Output Line #: 2

Set To: On

Refer to Security for more information.

The Configuration icon is available in all three (3) windows: CAD/CAM/CNC.

Click the icon to examine or modify FlashCut settings.



1. Menu selection tab
2. Menu selection tab
3. Configuration workspace
4. Configuration workspace

12.3 Menu Selection Tab

Select the appropriate tab to configure different FlashCut settings.

12.4 Setup Tab

When the Setup tab is selected, the setup file selections field and Setup Configuration Menu are visible.

12.5 Tooling Tab

When the Tooling tab is selected, the tool library file selection field and Tool Library are visible.

12.6 Setup File (*.setup extension)

The Setup File selection field is visible when the Setup tab is selected. The Setup File contains all of the configuration parameters for you machine tool. You can change the parameters using commands in the Setup Configuration Menu.

When FlashCut is first loaded, it prompts you to select a setup file. Setup files have the extension *.setup. Legacy files with the older extension *.STP can be loaded and updated. If you don't have a setup file to load, click Cancel to tell the program to load a default configuration, which you may edit and save under a different file name. Each setting is described by category, along with a list of sub-categories. Clicking the plus-minus icons next o each subcategory enables you to toggle whether or not the list is displayed.

The panel at the top of the configuration window enables you to save a certain configuration in the .setting file format.

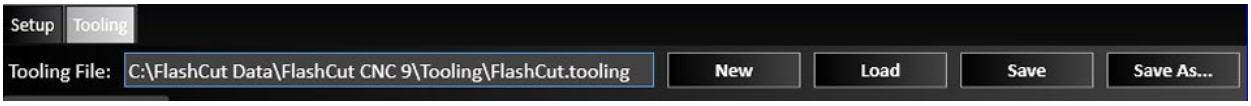
The New button allows you to duplicate a previously-created setup file. Select the file to be copied then click OK. Note that this is a temporary copy of the file that must be saved in order to solidify any changes; it will not affect the original file.

The Load button prompts you to save the current configuration setup and then loads a previously STP file.

Click Save to save the current configuration state. By default, FlashCut sores setup files in the C:\ProgramFiles\FlashCut CNC 8\Application Data\Configuration folder but you may not have sufficient privilege to store your setup files there. Choose an alternative location.

If a file has not been created yet, the Save As… button enables you to create a new .setup file and destination.

12.7 Tooling File (extension *.tooling)



The Tooling file contains all of the tool library parameters for your machine tool. You can change for parameters in the Tool Library.

When FlashCut is first loaded, it prompts you to select a tooling file. Tooling files have the extension *.tooling.

The panel at the top of the configuration window enables you to save a certain configuration in the tooling file format.

The New button allows you to duplicate a previously-created tooling file. Select the file to be copied then click OK. Note that this is temporary copy of the file that must be saved in order to solidify any changes; it will not affect the original file.

The Load button prompts you to save the current configuration and then loads a previously-created file.

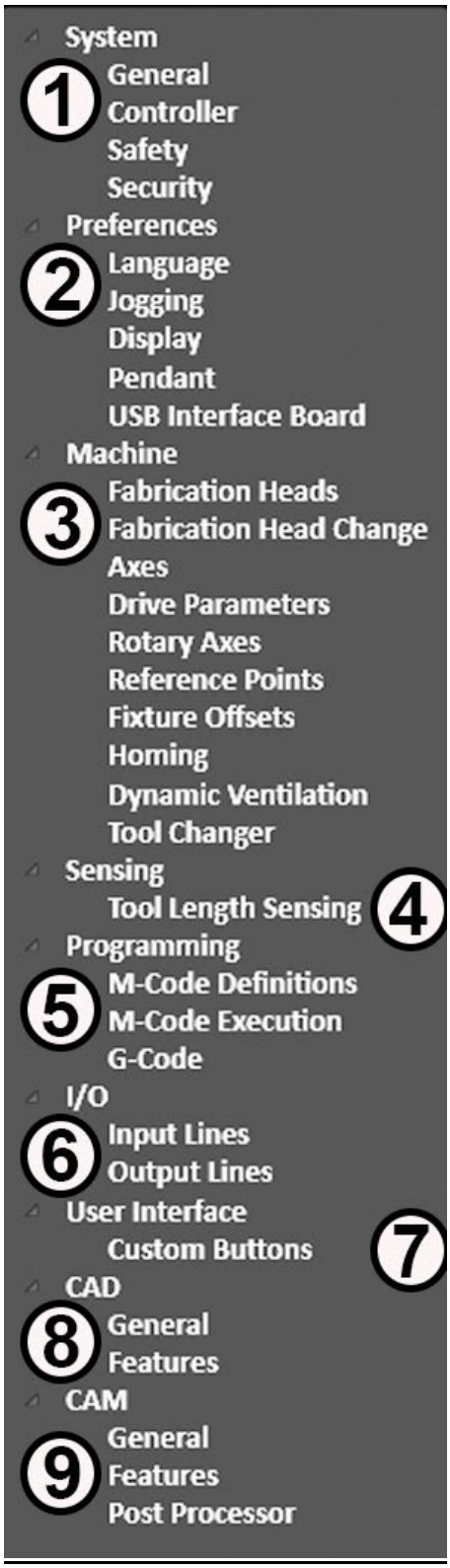
Click Save to save the current configuration state. By default, FlashCut stores setup files in the C:\ProgramFiles\FlashCut CNC 8\Application Data\Tooling folder but you may not have sufficient privilege to store your setup files there. Choose an alternative location.

If a file has not been created yet, the Save As…. button enables you to create a new .tooling file and destination.

12.8 Configuration Workspace

Detailed configuration options for each menu item are displayed in the configuration workspace.

12.9 Setup Configuration Menu

1.	System • General • Controller • Safety • Security	
2.	Preferences • Language • Jogging • Display • Pendant • USB Interface Board	
3.	Machine • Fabrication Heads • Fabrication Head Change • Axes • Drive Parameters • Rotary Axes • Reference Points • Fixture Offsets • Homing • Dynamic Ventilation • Tool Changer	
4.	Sensing • Tool Length Sensing	

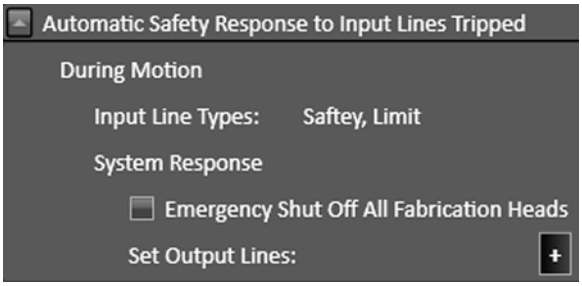
5.	Programming • M-Code Definitions • M-Code Execution • G-Code	
6.	I/O • Input Lines • Output Lines	
7.	User Interface • Custom Buttons	
8.	CAD • General • Features	
9.	CAM • General • Features • Post Processor	

12.10 System

- General
- Controller
- Safety

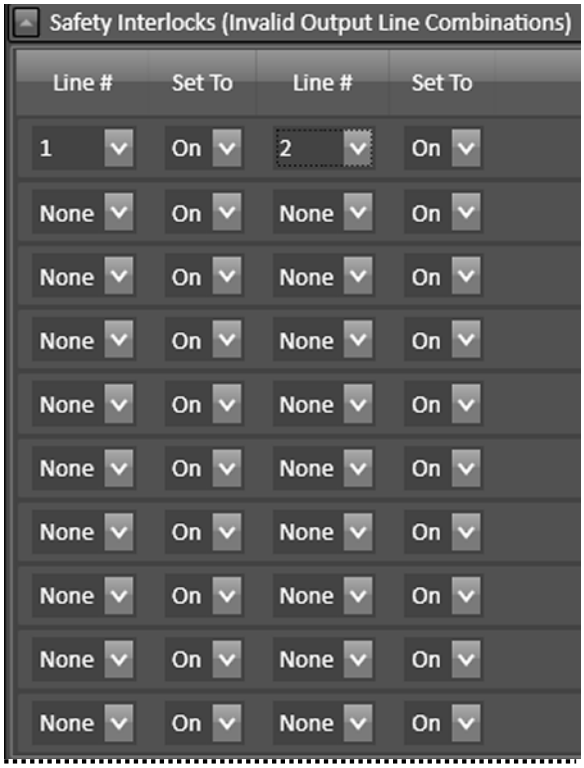
12.11 Automatic Safety Response

When selected, all fabrication heads will shut off when a safety or limit input line is tripped.



12.12 Safety Interlocks

The Safety Interlocks table lets you define incompatible combinations of output line on/off states. FlashCut will prevent the output lines from reaching a state that violates any of the safety interlocks you define.



To create an interlock, set the four (4) pulldown menus across a row of the table to define the incompatible states of two (2) output lines.

Example: The machine tool has an automatic tool changer that uses a spindle with a pneumatic tool chuck and the chuck should never be open when the spindle is turning. Assuming the spindle is on when output line 1 is on and the chuck is open when output line 2 is on, the four (4) pulldown menus on the first line of the table would be set as shown:

Output Line #: 1

Set To: On

Output Line #: 2

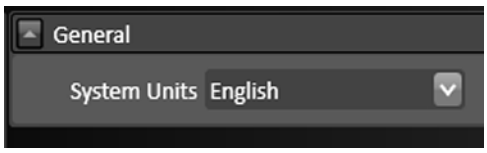
Set To: ON

- Security

12.13 General

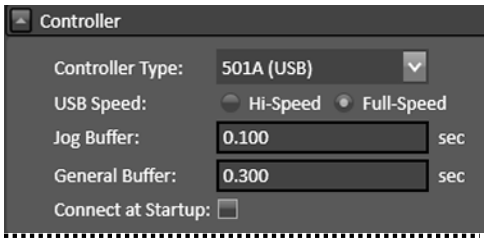
12.13.1 System Units

Use the System Units dropdown to choose which units of measurement to use. You may choose:



- English
- Metric

12.13.2 Controller



The Controller settings section presents the following controls:

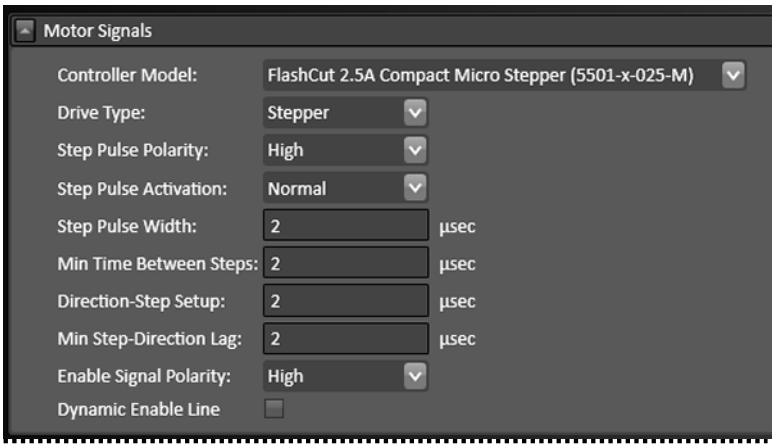
- Controller Type
- USB Speed
- Jog Buffer (sec)
- General Buffer (sec)
- Connect at Startup

1. Controller Type

   Specifies the type of signal generator (controller) you are using:
   501A (USB connection)
2. USB Speed
   USB Speed allows you to specify the speed of USB communications with the signal generator. Setting the USB speed to Full Speed helps to prevent communication problems.
3. Jog Buffer (sec)
   The jog generator buffer prevents events (such as screen updates) from affecting motor movement on the machine tool. The larger the buffer, the less effect system events have on motor movement. The smaller the buffer, the more responsive the machine tool is to mouse clicks and keyboard commands. In most cases, the lag-time between the PC and the motor movements is imperceptible. The value can range from 0.01 to 1.0 seconds. Slower computers may require a higher value.
   The jog buffer setting sets the buffer size on the signal generator for jogging in continuous mode. Typically, this value is lower than the general value, to make the jog buttons more responsive.
4. General Buffer (sec)
   General Buffer (sec) sets the buffer size on the signal generator for all operations except jogging in continuous mode.
5. Connect at Startup

   When this option is selected, FlashCut will attempt to connect to the default machine controller when the program launches.

12.14 Motor Signals



- Driver Model
- Driver Type
- Step Pulse Polarity
- Step Pulse Activation
- Step Pulse Width (µsec)
- Min Time Between Steps (µsec)
- Direction-Step Setup (µsec)
- Min Step Direction Lag (µsec)
- Enable Signal Polarity
- Dynamic Enable Line

1. **Driver Model**

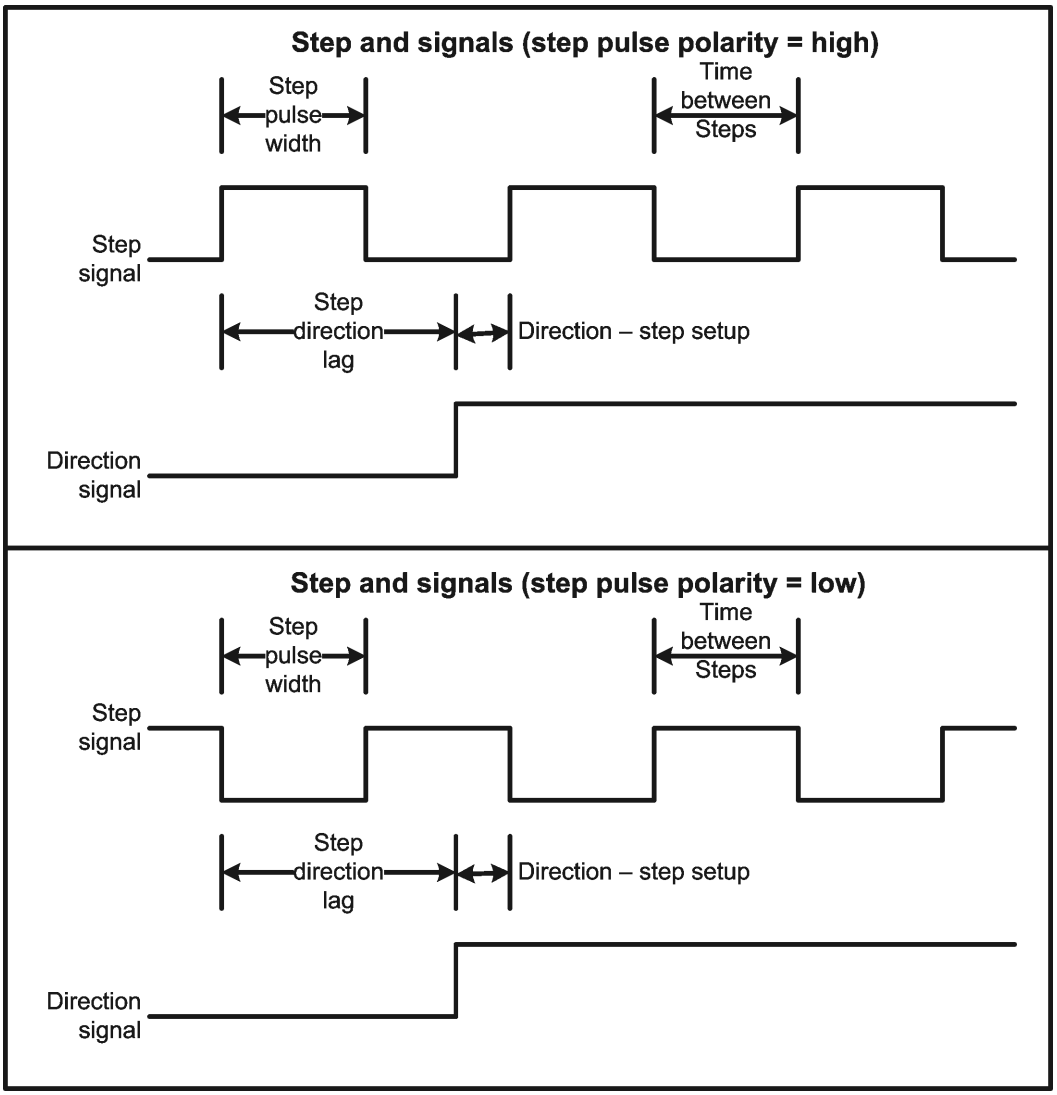
   Set the model of your motor driver or combination controller/driver box.

   If you are using a FlashCut driver or combination controller/driver box, the model number is printed on a sticker located on the back or bottom of the box. When you select your driver model, FlashCut enters the correct values for all other signal settings (Step Pulse Polarity through Dynamic Enable Line). The fields are disabled and cannot be changed. If you need to change the values, choose Other (generally not necessary).

   If you are using non-FlashCut drivers, choose Other. This enables all fields, allowing you to enter the proper settings based on your motor driver requirements. Note that same signal fields do not apply to all signal generator models and may be disabled based on your controller settings (signal generator model and chip).

2. **Driver Type**
3. **Indicates your motor driver type: stepper or servo.**
4. **Step Pulse Polarity**

   Sets the polarity of the stop signal to High or Low as shown in this diagram.



5. Stop Pulse Activation

(Model 401A Signal Generator Only) Indicates how the signal generator creates step pulses as follows:

Normal – The signal generator times the duration of each step pulse.

Inverted – The signal generator times the delay between step pulses, while guaranteeing the step pulse duration is at least the step pulse width entered. When the motor driver requires a long step pulse, this mode yields higher step rates.

Optimize – FlashCut automatically selects the best activation mode.

6. Step Pulse Width (µsec)

Sets the duration of the step pulse (in microseconds). See the diagram above.

7. **Min Time Between Steps (µsec)**

   Sets the minimum time between step pulses (in microseconds). See the diagram above.

8. **Direction-Step Setup (µsec)**

   Sets the time between a direction change and the leading edge of the subsequent step pulse. (in microseconds). See the diagram above.

9. **Min Step Direction Lag (µsec)**

   Sets the minimum time from the leading edge of a step pulse to a subsequent direction change (in microseconds). See the diagram above.

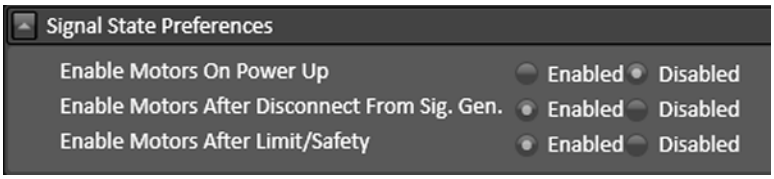
10. **Enable Signal Polarity**

    Sets the polarity of the enable signal. Choose High if the motor driver is enabled by a high signal, or Low if the motor driver is enabled by a low signal.

11. **Dynamic Enable Line**

    Sets the behavior of the enable line. When checked, the enable line turns on when any axis is in motion and turns off when all axes are idle. Generally this box should be unchecked.

12.15 Signal State Preferences



- Enable Motors On Power Up
- Enable Motors After Disconnect From Sig. Gen.
- Enable Motors After Limit/Safety

1. **Enable Motors On Power Up**

   (Model 501A signal generator only) Sets the state to which the signal generator drives the motor enable line when turned on, to initially enable or disable the motors.

2. **Enable Motors After Disconnect From Sig. Gen.**

   Sets the state to which the signal generator drives the motor enable line when you disconnect from the signal generator.

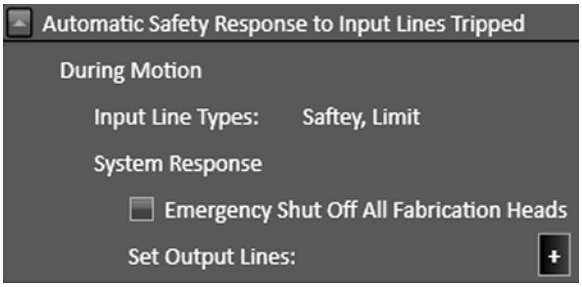
3. **Enable Motors After Limit/Safety**

Sets the state to which the signal generator drives the motor enable line after a limit or safety error. See Line Setup for more information on these errors.

12.16 Safety

1. **Automatic Safety Response**

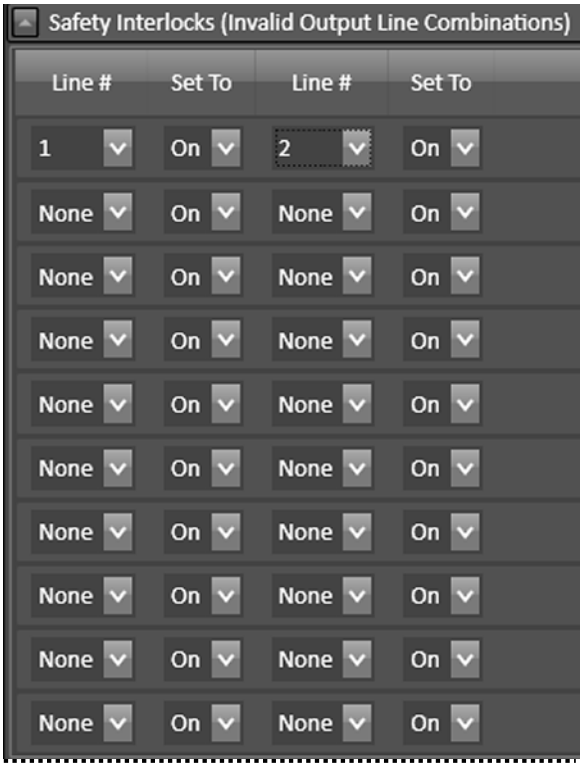
   When selected, all fabrication heads will shut off when on safety or limit input line is tripped.



2. **Safety Interlocks**

   The Safety Interlocks table lets you define incompatible combinations of output line on/off states. FlashCut will prevent the output lines from reaching a state that violates any of the safety interlocks you define.

   To create an interlock, set the four (4) pulldown menus across a row of the table to define the incompatible states of two (2) output lines.



   Example: The machine tool has an automatic tool changer that users a spindle with a pneumatic tool chuck and the chuck should never be open when the spindle is turning. Assuming the spindle is on when output line 1 is on and the chuck is open when output line 2 is on the four (4) pulldown menus on the first line of the table would be set as shown.

   Output Line #: 1

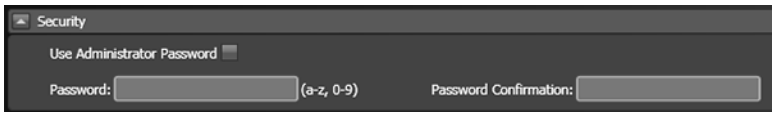
   Set To: On

   Output Line #: 2

   Set To: On

12.17 Security

- Use Administrator Password
- Password and Password Confirmation



1. **Use Administrator Password**

   Check this box to set or change the Administrator Password. Enter the desired

2. **Password and Password Confirmation**

   Both fields must match to confirm the password. Passwords may contain only letters or digits and be between 1-15 characters in length.

12.18 Preferences

- Language
- Jogging
- Display
- Pendant
- USB Interface Board

12.19 Language

1. **Preferred Language**

Use the Languages configuration option to choose the language FlashCut uses for its user interface. You may choose from:

- English (United States)
- Spanish (Spain)
- Italian (Italy)

- French (France)
- Portuguese (Portugal)

2. **Jogging**

   The Jogging configuration option displays the following controls:

   - Jog and Point Move Rates
   - Jog Distances
   - General

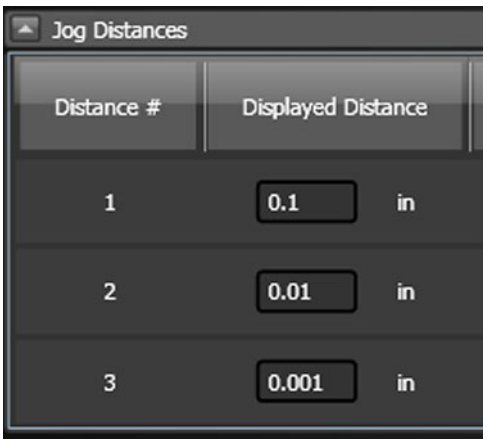
3. **Jog and Point Move Rates**

   Use the Jog Rates controls to set the default slow jog and pint move rates for each axis.

   Slow

   This column corresponds to the feed rate specified in the Jog and point control panel. FlashCut executes all jog moves using the rates you enter in these fields.

12.20 Jog Distances



1. **Distance #**

   The distance number identifies which button on the main screen the displayed distance refers to.

2. **Displayed Distance**

   This is the value of the incremental move that is commanded each time the user clicks a directional jog button on the CNC panel.

12.21 General

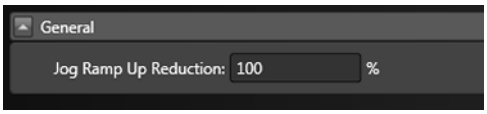
Jog Ramp Up Reduction

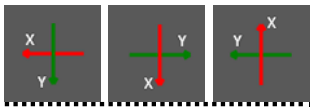
Set the percentage of the speed at which the rate increases during jog operations. The default is 100%.

12.22 Display

1. **Axis Display**

   Check the appropriate check box for the desired axis settings. The icon displayed will change depending on the settings selected:



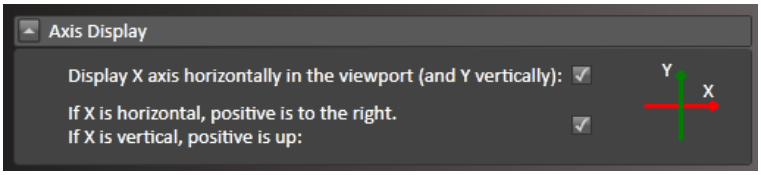


2. **Current Position**

   Select the desired icon to mark the current position.

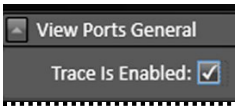
3. **View Ports General**

   Select to enable Trace in viewports.



4. **Pendant**

   The Flex Pendant is a versatile accessory that you can customize to your application. You can assign functions to each button to create your own custom layout.



5. **General**

   - **Use Pendant**

     Enables control of the machine from the pendant.

   - **Number of Jog Rates**

     Sets the number of jog rates (2-12) that can be defined.

   - **Model**

     Current models include 20-key, 24-key, and 12-key with joystick.

   - **Pendant Buttons**

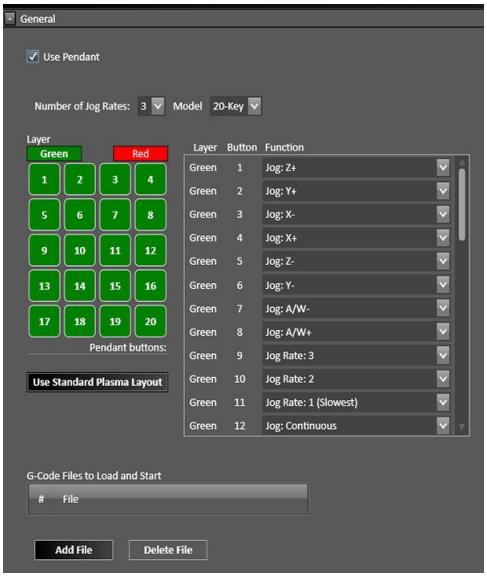
     Two (2) sets of buttons (Green and Red) can be configured. Click the button you wish to configure and then assign its function.

6. **Use Standard Plasma Layout**

   Selecting this button will load the pre-defined settings. All current custom function settings will be overwritten.

7. **Functions**

   - Layer (green or red)
   - Button (#)
   - Function (select from dropdown menu)



8. **G-Code Files to Load and Start**

   Up to ten (10) different G-Code files can be entered into this table. Use the Add File and Delete File buttons to add or remove files from the list.

   The function Load G-Code and Start File # can then be assigned to a button on the pendant. When the button is pressed, the given file will be loaded and run.

12.23 USB Interface Board

A USB Interface Board is available from FlashCut for some machines. The board allows direct control of the machine by assigning functions to different switches. The functions of the board can be programmed here.

12.24 General

1. **Use USB Interface Board**

   Enables control of the machine from the board.

2. **Number of Jog Rates**

Sets the number of jog rates (2-12) that can be defined.

3. **Functions**

   Use the Add Function and Delete Function buttons to define or remove the functions assigned to different row, column, and layer combinations.

   Each function has four (4) attributes:

   - Layer (green or red)
   - Wiring Row (1-8)
   - Wiring Column (1-12)
   - Function (select from dropdown menu)

4. **File Table**

   Up to ten (10) different G-Code files can be entered into this table. Use the Add File and Delete File buttons to add or remove files from the list.

   The function Load G-Code and Start File # can then be assigned to a button on the pendant. When the function is executed, the given file will be loaded and run.

12.25 Machine

- Fabrication Heads
- Fabrication Head Change
- Axes
- Drive Parameters
- Rotary Axes
- Reference Points
- Homing
- Dynamic Ventilation
- Sensing

12.26 Fabrication Heads

This section explains in detail how to configure each of the different types of Fab Heads used by FlashCut.

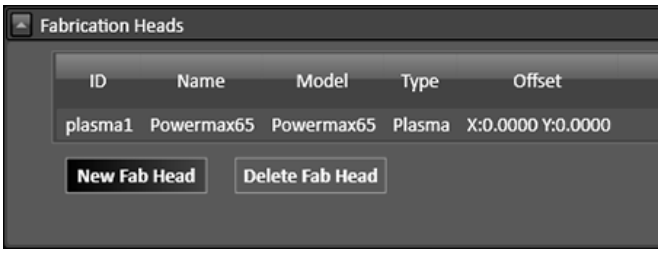
- Fabrication Heads
- Cut Charts

1. **Fabrication Heads**

   Fabrication heads are the actual cutting head on your machine.

   You can have more than one installed on your machine. Types include:

   - Plasma
   - Oxyfuel
   - Waterjet
   - Laser
   - Laser Pointer
   - Marker
   - Mill
   - Tangential Knife
   - Hotwire



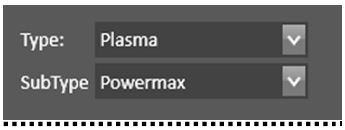
2. Each type of Fabrication Head (or Fab Head) has its unique settings. In this section, you can edit an existing Fab Head by selecting it from the list or you can delete or create new one.

   To edit a Fab Head, select it from the list and the Fab Head parameters will appear.

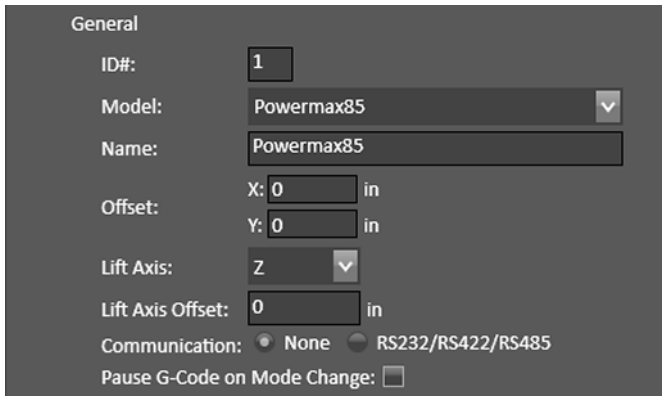
   To create a new Fab Head, select the New Fab Head button. To delete an existing Fab Head, first select its name in the list and then select the Delete Fab Head button.

   The parameters for the various types of Fab Heads are outlined below.

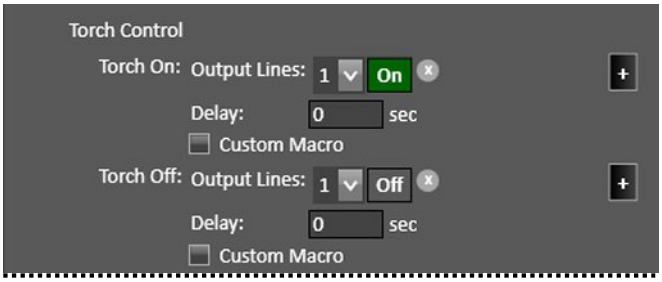
3. **Plasma**



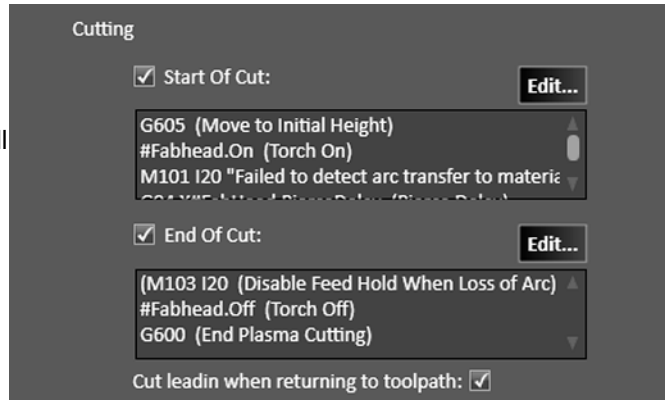
- **Type and Sub Type** – When Plasma is selected type, a dropdown list of subtypes will be displayed.
- **ID and Name** – These values will be automatically populated, but you can edit them.
- **Model** – Select from the dropdown list.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.

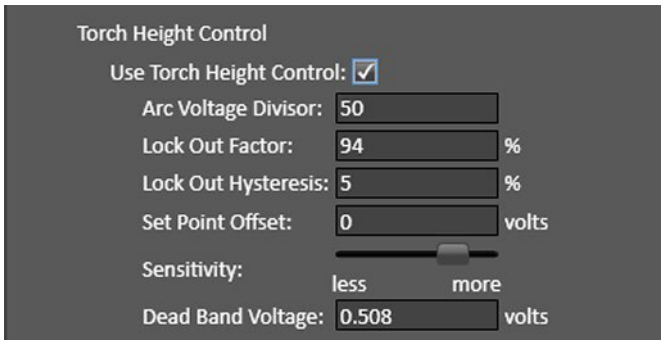


- **Lift Axis Offset** – Enter offset for selected lift axis.
- **Communication** – Select RS-232/RS-485 and a Comm port if you have direct communication with the plasma for setting Current and Pressure on the fly.
- **Pause G-Code on Mode Change** – Select to enable.
- **Torch Control**
- **Torch On/Off Output Lines** – When selected, define output line(s) used to control the Fab head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.



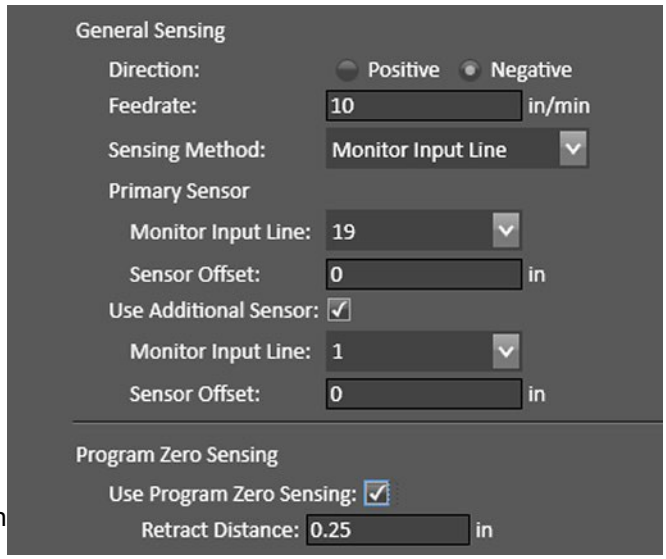
- **Custom Macro** – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.
- **Cutting**



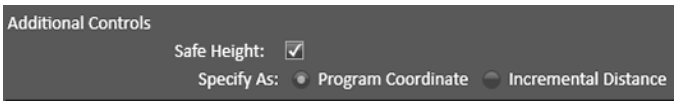


- **Start/End of Cut** – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.
- **Cut lead in when returning to toolpaths** – Clerk to allow recovery from a lost torch.
- **Torch Height Control**
- **Use Torch Height Control** – Select this value to enable torch height control if your torch has it.
- **Arc Voltage Divisor** – This is typically 50 and must match the exact voltage divider setting in your torch. Note that in order for the torch height control system to work properly, (maximum raw voltage)/(arc voltage divisor) must be smaller than 6V.
- **Lock Out Factor** – Turns the THC off to prevent unwanted diving when the XY velocity of the torch is below this percentage of the programmed feedrate.
- **Lock out Hysteresis** – The THC will turn back on again when the XY velocity goes above the ((Lock Out Factor) + (Lock Out Hysteresis))* (Programmed Feedrate).
- **Set Point Offset** – Offsets the small residual voltage when torch is not in use.
- **Sensitivity** – Determines how sensitive the THC acts towards voltage fluctuation. When the slider bar is adjusted, the new value appears in the Dead Band Voltage window below.
- **General Sensing**
- **Direction** – Select direction.
- **Feedrate** – Enter value in in/min.
- **Sensing Method** – Select either Monitor Input Line or Monitor Input Current.
- **Primary Sensor** – The Input Line that will be used for the primary sensor. This is typically an Ohmic sensor.
- **Sensor Offset** – The extra distance that the Z-Axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch or Z-Axis mechanical system.
- **Use Additional Sensor** – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

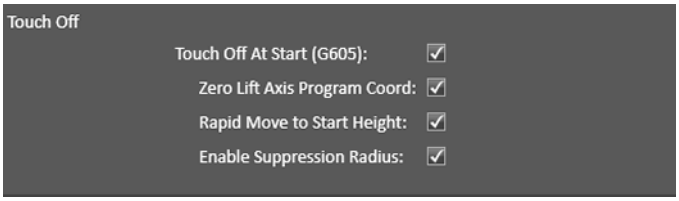
- **Program Zero Sensing**
- **Use Program Zero Sensing** – Check the box for the ability to seek the sheet surface to set program zero.
- **Retract Distance** – The amount that the Fab Head will move up after sensing the sheet.
- **Additional Controls** – Check the box to enable a control. Specific values for these controls are set in the CNC window.



- **Safe Height** – This value can be specified as Program Coordinate or Incremental Distance.
- **Touch Off** – Check the box to enable a control.



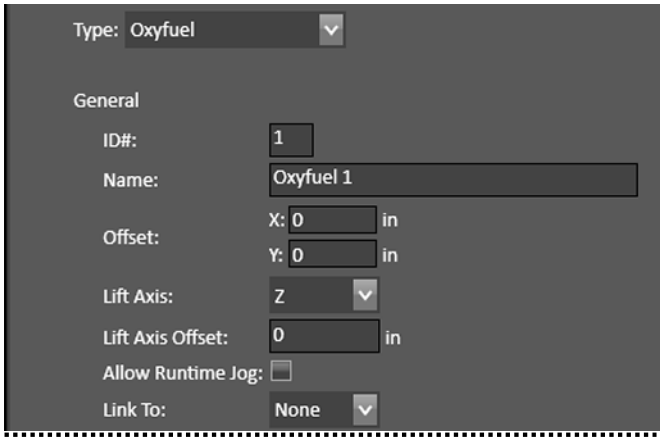
- **Touch Off at Start (G605)** – Select to sense the sheet at the start of each cut.
- **Zero Lift Axis Program Coord** – When selected, laser will zero lift axis program coordinate on every touch off.



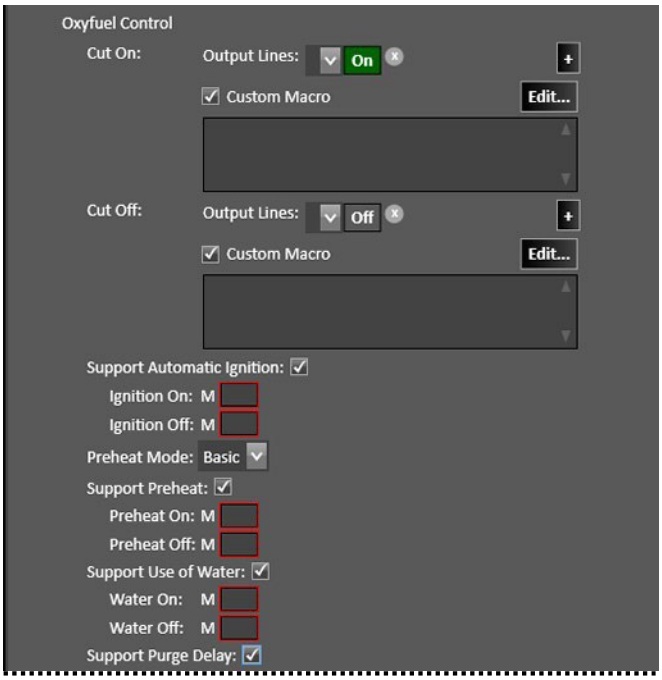
- **Rapid Move to Start Height** – Select to enable rapid moves.
- **Enable Suppression Radius** – Setting a suppression radius prevents additional touch-offs within the specified difference from the first touch off point.

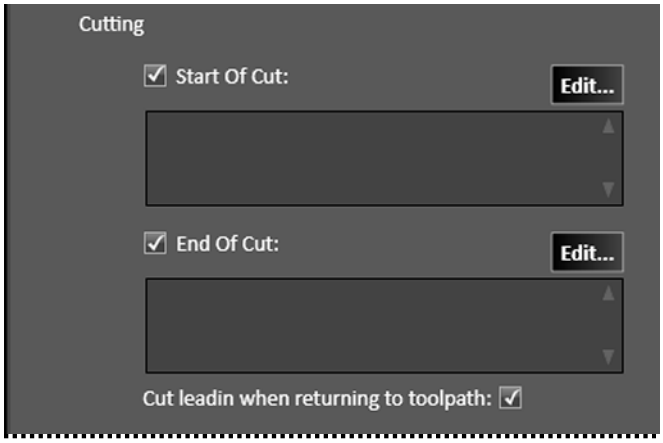
12.27 Oxyfuel

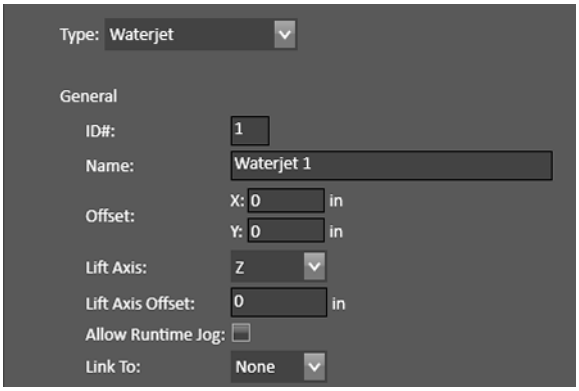
- **ID and Name** – These values will be automatically populated, but you can edit them.



- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.
- **Link To** – Allows for the use of multiple simultaneous Fab Heads. Available heads will appear in the dropdown menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.
- **Oxyfuel Control**
- **Cut On/Off Output Lines** – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.







Custom Macro – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.

Support Automatic Ignition – Enables this feature. M-Code required.

Preheat Mode – Basic or Bi-Level may be selected. Note that for Bi-Level modes, two (2) sets of M-Code values must be defined. Missing fields will be flagged in red as shown.

Support Use of Water – Enables this feature. M-Codes required for Water On/Off.

Purge Delay – Enables this feature.

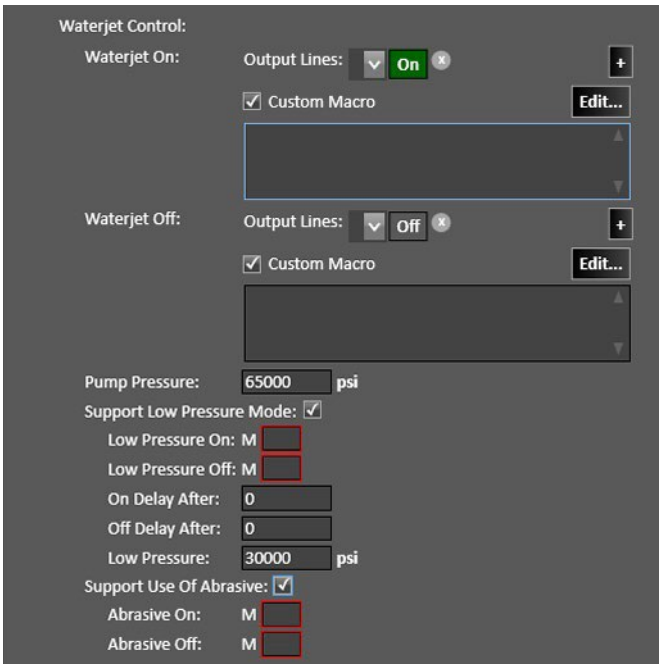
- **Cutting**

  Start/End of Cut – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.

- **Cut lead in when returning to toolpath** – Check to allow recovery from a lost torch.

12.28 Waterjet

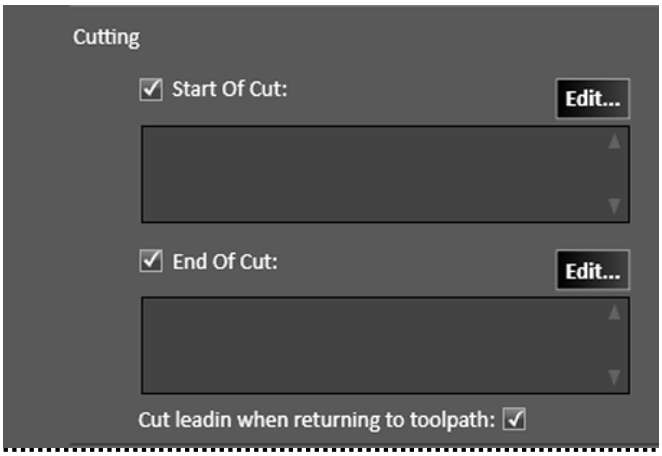
When defining a new Waterjet head, select the Type from the dropdown lists.



- **ID and Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.
- **Link To** – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the dropdown menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.
- **Waterjet Control**

  Waterjet On/Off Output Lines – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

Custom Macro – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.



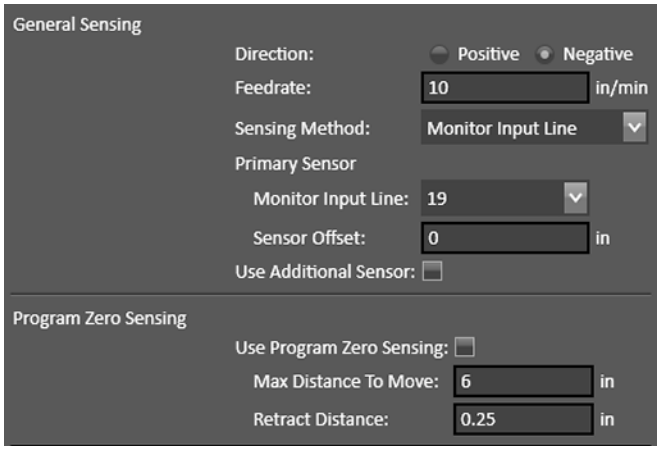
On/Off Delay After – Time in seconds.

Low Pressure – When low pressure mode is active, enter the pressure in PSI here.

- **Support Use of Abrasive** – When checked, enter the proper M-Codes for turning flow of abrasive compound on and off. Note that the M-Code must be defined in the M-Code Definitions.
- **Cutting**

  Start/End of Cut – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.

  Cut lead in when returning to toolpath – Check to allow recovery from a lost jet.



- **General Sensing**

  Direction – Select direction.

  Feedrate – Enter value in units/min.

  Sensing Method – Select either Monitor Input Line or Monitor Input Current

  Primary Sensor – The Input Line that will be used for the primary sensor. This is typically an Ohmic sensor.

  Sensor Offset – The extra distance that the Z-Axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch or Z-Axis mechanical system.

Use Additional Sensor – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

- **Program Zero Sensing**

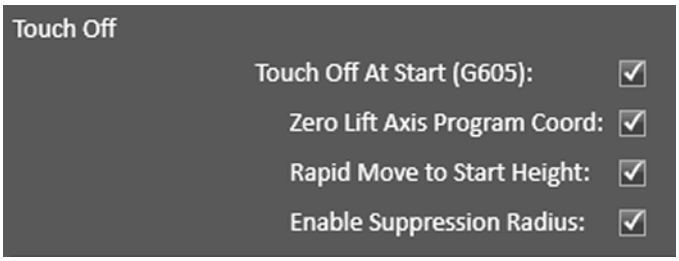
  Use Program Zero Sensing – Check the box for the ability to seek the sheet surface to set program zero.

  Max Distance to Move – The maximum travel distance to find the sheet.

  Retract Distance – The amount that the Fab Head will move up after sensing the sheet.

- **Touch Off** – Check the box to enable a control.

  Touch Off at Start (G605) – Select to sense the sheet at the start of each cut.



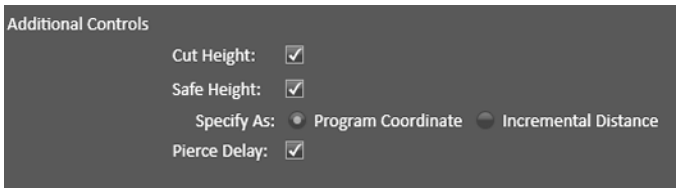
  Zero Lift Axis Program Coord – When selected, laser will zero lift axis program coordinate on every touch off.

  Rapid Move to Start Height – Select to enable rapid moves.

  Enable Suppression Radius – Setting a suppression radius prevents additional touch-offs within the specified difference from the first touch off point.

- **Additional Controls** – Check the box to enable a control.

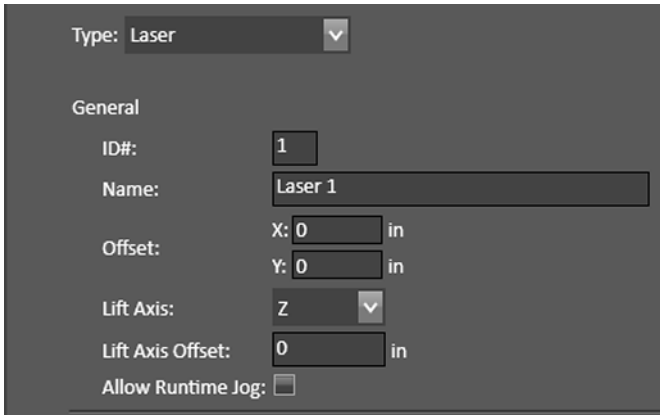
  Cut Height – Select to enable.



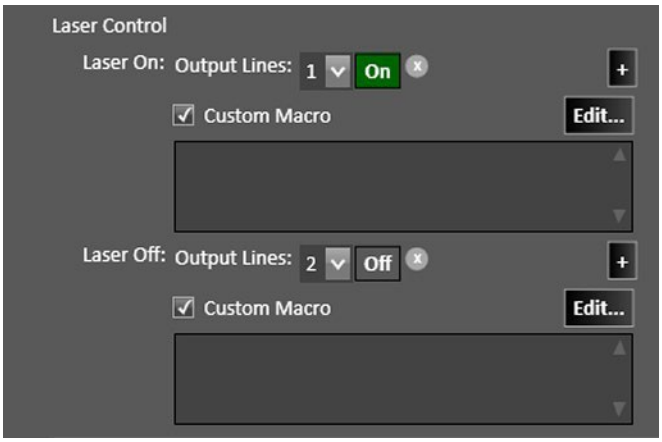
  Safe Height – This value can be specified as Program Coordinate or Incremental Distance.

  Pierce Delay – Select to enable.

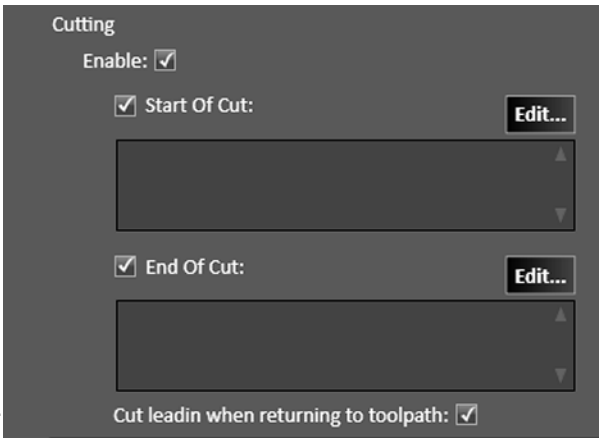
12.29 Laser



- **ID and Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.



- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.
- **Link To** – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the dropdown menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.



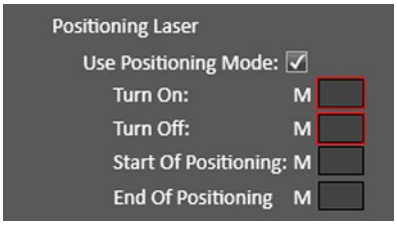
- **Laser Control**

  Laser On/Off Output Lines – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

- **Cutting**

  Start/End of Cut – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.

  Cut lead in when returning to toolpath – Check to allow recovery from a lost torch.



- **Marking**

  Start/End of Mark – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.

- **Positioning Laser**

  Use Positioning Mode – Enable this feature.

  Turn On/Off – M-Codes required.

  Start/End of Positioning – M-Codes required.

- **Height Control**

Use Height Control – Select to enable.

Voltage Scale Factor – Number

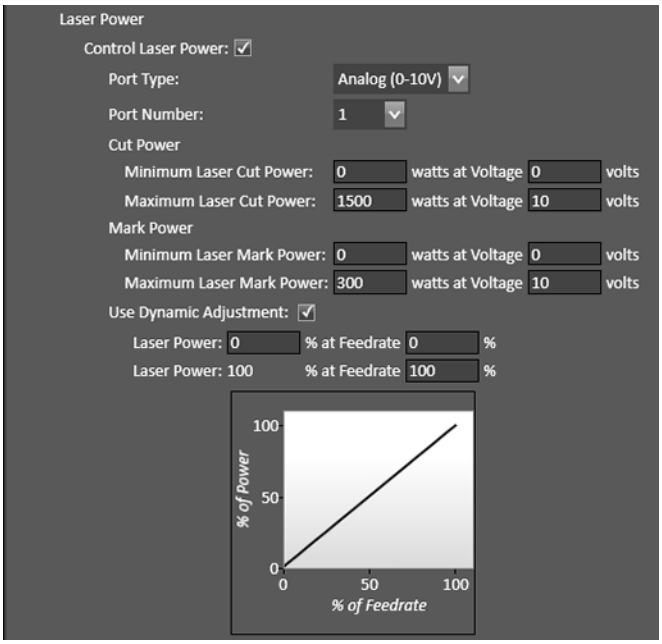
Lock Out Factor – Percentage

Lock Out Hysteresis – Percentage

Idle Voltage Offset – Select to enable.

Sensitivity – Slider control sensitivity.

Dead Band Voltage – Value calculated automatically from Sensitivity.



Height Control Calibration – Enter M-Code.

Calibration Output Line – Enter M-Code.

Pre Collision Offset – Enter distance.

- **Measuring Range**

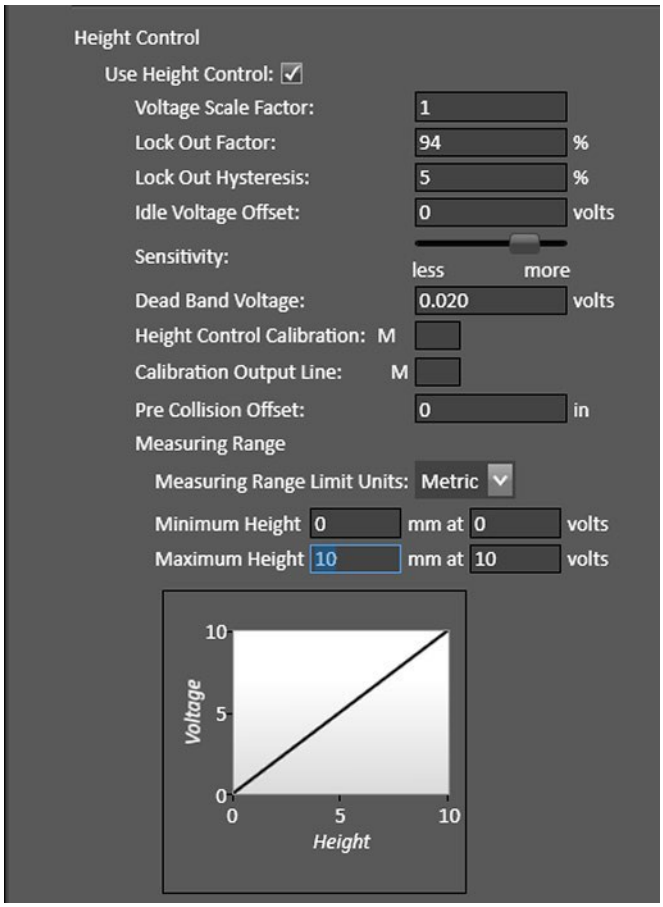
  Measuring Range Limit Units – Select English or metric.

  Minimum/Maximum Height – Enter distance and voltage.

- **Laser Power**

  Control Laser Power – When checked, allows laser cutter output to be configured and controlled more precisely.

  Port Type – Options are Analog, Pulse Width Modulation, and Pulse Frequency Modulation.



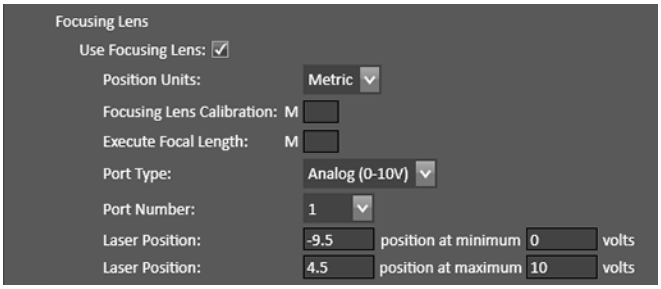
Port Type – Select Analog, Pulse-Width Modulation (PWM), or Pulse-Frequency Modulation (PFM).

For Analog, select Port Number. For PWM, enter Frequency. For PFM, enter Duty Cycle.

Minimum and Maximum Laser Power – Enter watts, voltage.

Use Dynamic Adjustment – Alters laser power based on feedrate. The slope of the curve changes as the values for laser power and feedrate change.

Focusing Lens



Use Focusing Lens – Select to enable.

Focusing Lens Calibration – Enter M-Code.

Execute Focal Length – Enter M-Code.

Port Type – Select Analog, Pulse-Width Modulation (PWM), or Pulse-Frequency Modulation (PFM).

For Analog, select Port Number. For PWM, enter Frequency. For PFM, enter Duty Cycle.

Laser Position at Minimum/Maximum Volts – Enter distance and voltage.

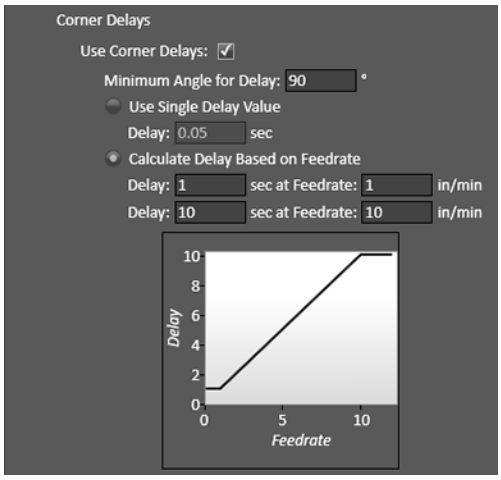
- **Corner Delays**

  User Corner Delays – When checked, the cutter will pause at corners during an operation.

  Minimum Angle for Delay – Set the minimum corner angle that will trigger a delay.

  Use Single Delay Value – The delay can be an absolute value in seconds or a function of feed rate.

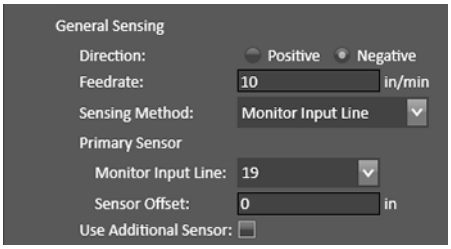
  Calculate Delay Based on Feed rate – By setting a range of values for delay time and feed rate, the timing of the delay over a range of feed rates can be set. The slope of the curve changes as the values change.



- **General Sensing**

  Direction – Select direction.

Feed rates – Enter value in in/min.



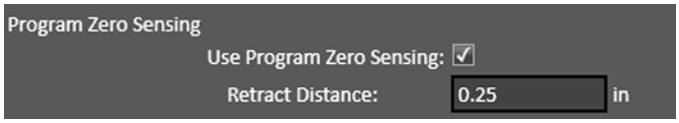
Sensing Method – Select either Monitor Input Line or Monitor Input Current.

Primary Sensor – The Input Line that will be used for the primary sensor. This is typically an Ohmic sensor.

Sensor Offset – The extra distance that the Z-Axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch or Z-Axis mechanical system.

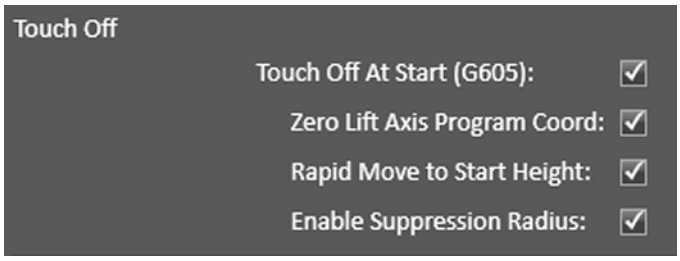
Use Additional Sensor – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

- **Program Zero Sensing**

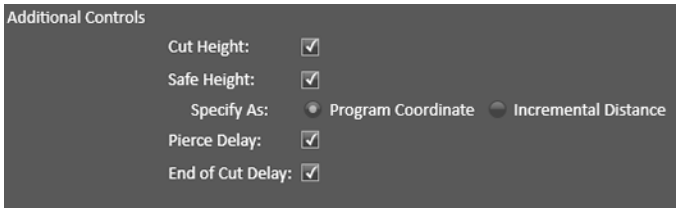


Use Program Zero Sensing – Check the box for the ability to seek the sheet surface to set program zero.

Retract Distance – The amount that the head will move up after sensing the sheet.

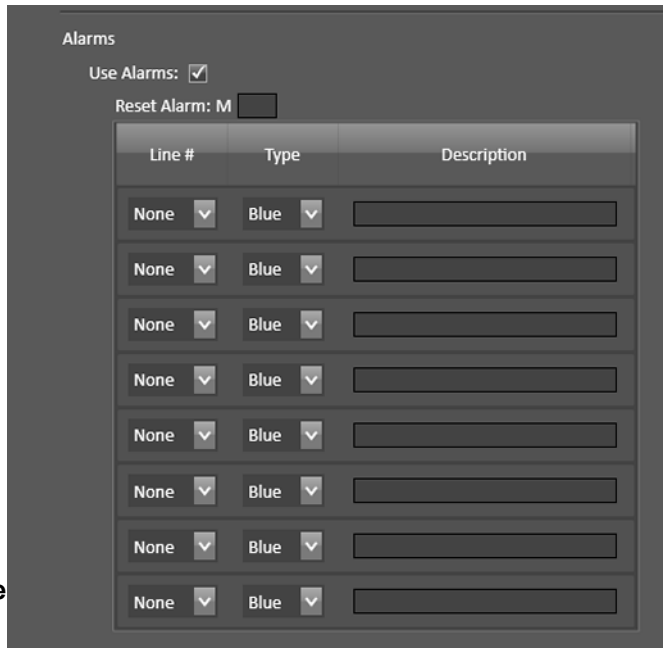


- **Touch Off** – Check the box to enable a control.



Touch Off at Start (G605) – Select to sense the sheet at the start of each cut.

Zero Lift Axis Program Coord – When selected, laser will zero lift axis program coordinate on every touch off.



Rapid Move to Start Height – Select to enable rapid moves.

Enable Suppression Radius – Setting a suppression radius prevents additional touch-offs within the specified difference from the first touch off point.

- **Additional Controls** – Check the box to enable a control.

  Cut Height – Select to enable.

  Safe Height – This value can be specified as **Program Coordinate** or **Incremental Distance Pierce Delay** – Select to enable.

  End of Cut Delay – Select to enable.

- **Alarms**

  Use Alarms – Select to enable.

  Reset Alarms – Enter M-Code.

  Line # - Select which input line to use for each alarm.

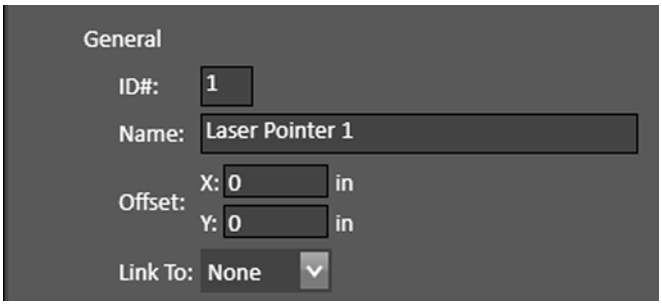
  Type – Color codes may be assigned to each alarm.

  Description – Enter text.

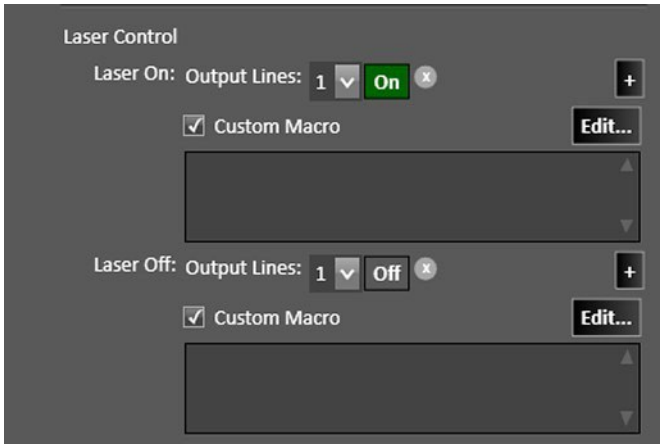
12.30 Laser Pointer

- **ID** and N**ame** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.

Link To – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the dropdown menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.



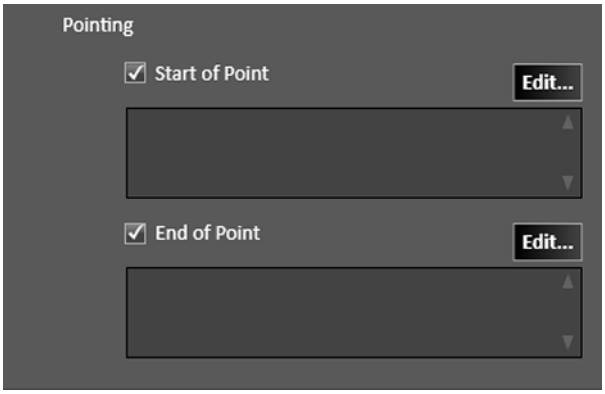
- **Laser Control**



Laser On/Off Output Lines – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

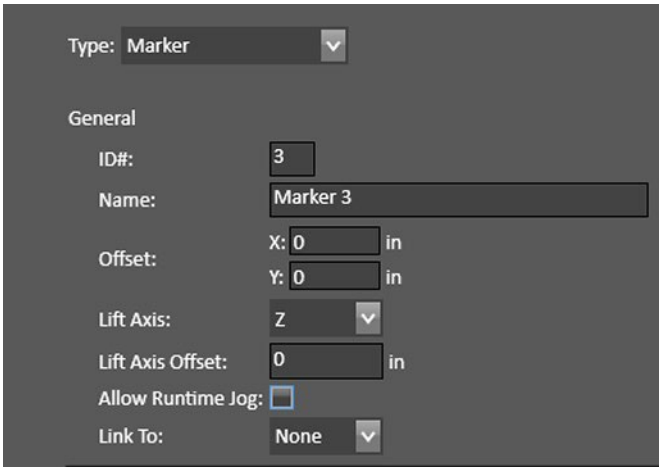
Custom Macro – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.

- **Pointing**

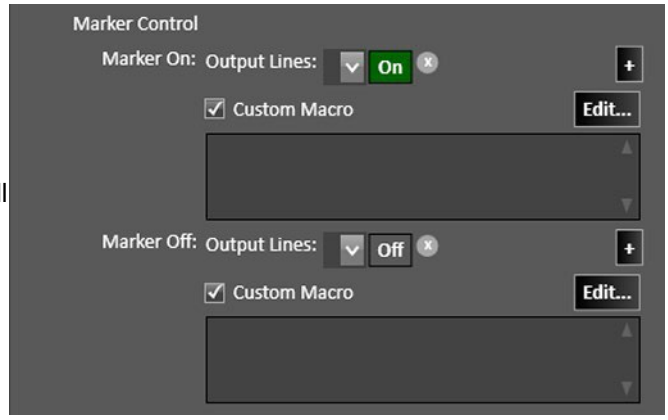


Start/End of Point – Select these fields to define operations that take place at the start or end of a point. Selecting Edit will bring up the code editor window.

12.31 Marker



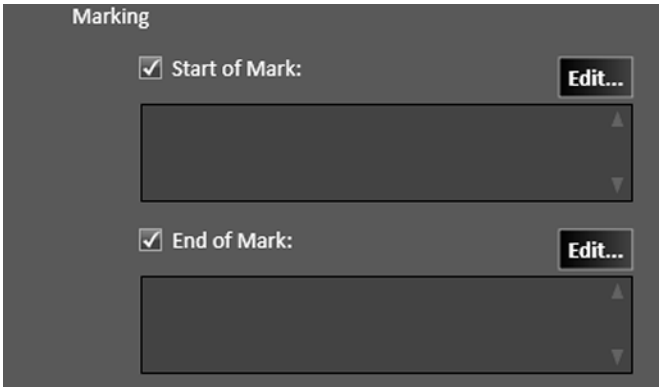
- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.
- **Link To** – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the dropdown menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.

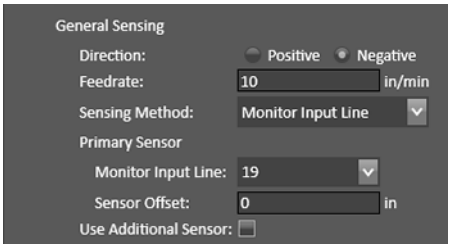


- **Marker Control**

  Marker On/Off Output Lines – When selected define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

- **Custom Marco** – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.
- **Marking**
- **Start/End of Mark** – Select these fields to define operations that take place at the start or end of a mark. Selecting Edit will bring up the code editor window.
- **General Sensing**





Direction – Select direction.

Feed rate – Enter value in in/min.

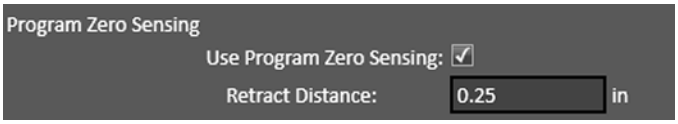
Sensing Method – Select either Monitor Input Line or Monitor Input Current.

Primary Sensor – The Input Line that will be used for the primary sensor. This is typically an Ohmic sensor.

Sensor Offset – The extra distance that the Z-Axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch Z-Axis mechanical system.

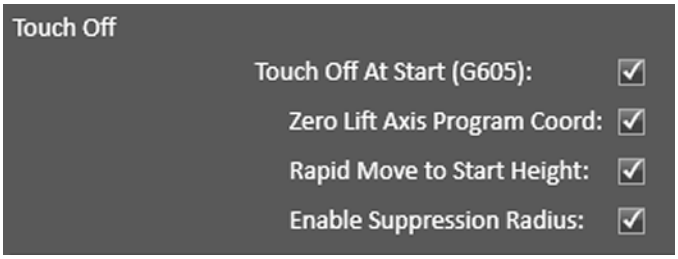
Use Additional Sensor – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

- **Program Zero Sensing**



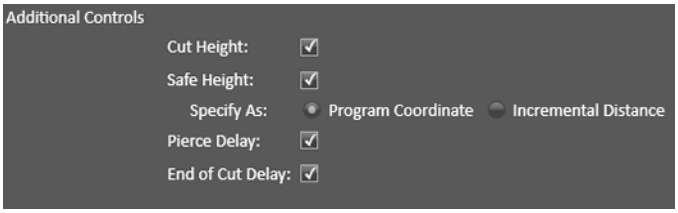
Use Program Zero Sensing – Check the box for the ability to seek the sheet surface to set program zero.

Retract Distance – The amount that the head will move up after sensing the sheet.



- **Touch Off –** Check the box to enable a control.

Touch Off at Start (G605) – Select to sense the sheet at the start of each cut.

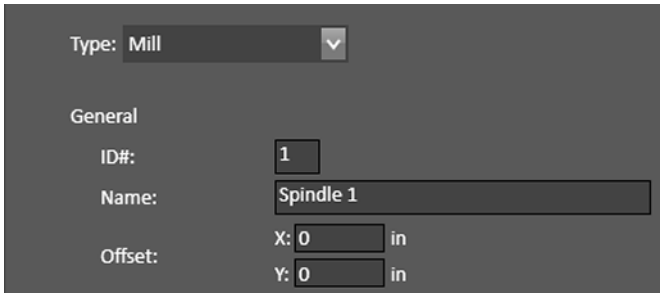


Zero Lift Axis Program Coord – When selected, Marker will zero lift axis program coordinate on every touch off.

- **Additional Controls**

Safe Height – This value can be specified as Program Coordinate or Incremental Distance.

12.32 Mill



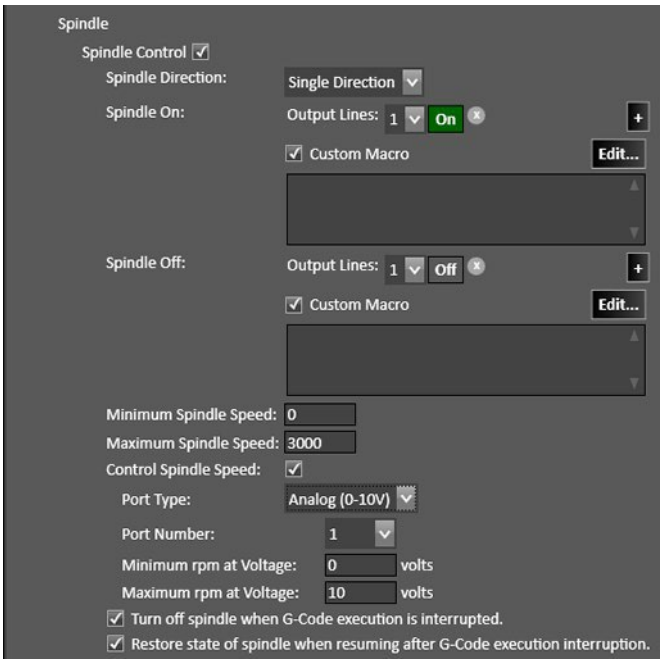
- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **General Sensing**

  Direction – Select direction.

  Feed rate – Enter value in in/min.

  Sensing Method – Select either Monitor Input Line or Monitor Input Current.

  If **Monitor Input Line** is selected, the following parameters apply:



- **Spindle**

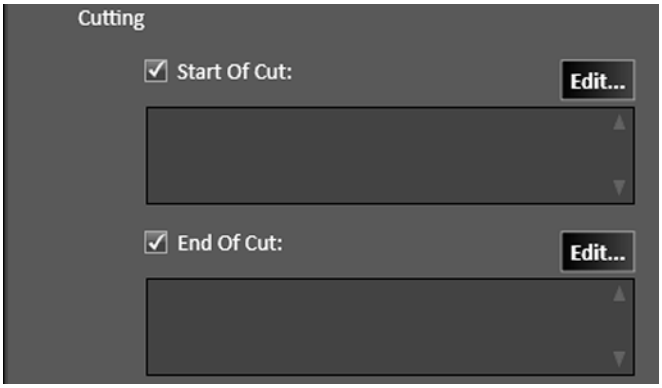
  Spindle Control – Select to enable.

  Spindle Direction – Select Single- or Bi-Direction.

  Spindle On/Off Output Lines – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

  If Bi-Direction, Spindle On controls can be configured for clockwise and counterclockwise (CW, CCW) operations.

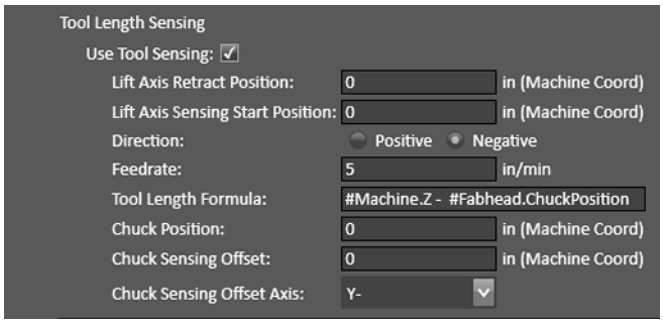
  Custom Macro – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting Edit will bring up the code editor window.



Minimum/Maximum Spindle Speed – Enter value.

Control Spindle Speed – Select to enable.

Port Type – Options are Analog, Pulse Width Modulation, and Pulse Frequency Modulation.

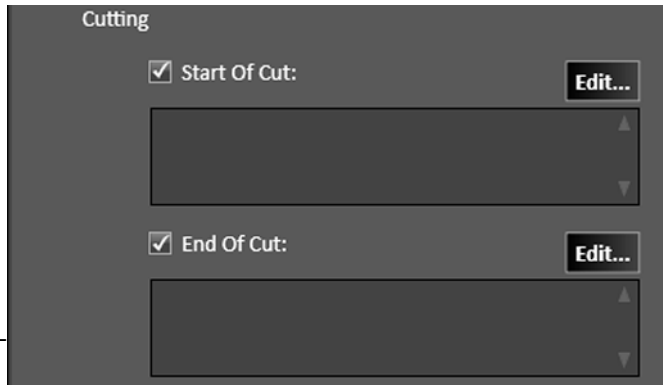


Port Type - Select **Analog, Pulse-Width Modulation (PWM), or Pulse-Frequency Modulation (PFM).**

For **Analog,** select **Port Number**. For **PWM**, enter **Frequency. PFM**, enter **Duty Cycle.**

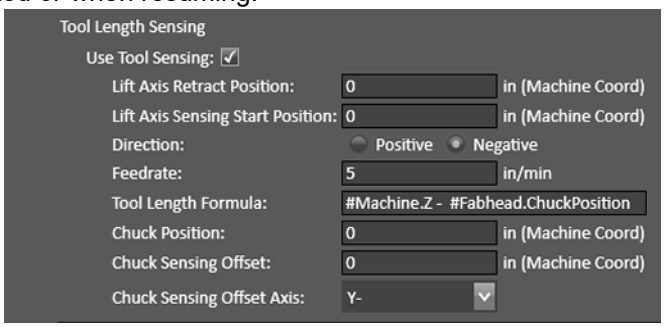
Minimum/Maximum RPM at Voltage – Range varies for different port types.

G-Code execution interruption – Select to configure spindle behavior when G-Code is interrupted or when resuming.



- **Cutting**

  Start/End of Cut – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.



- **Tool Length Sensing**

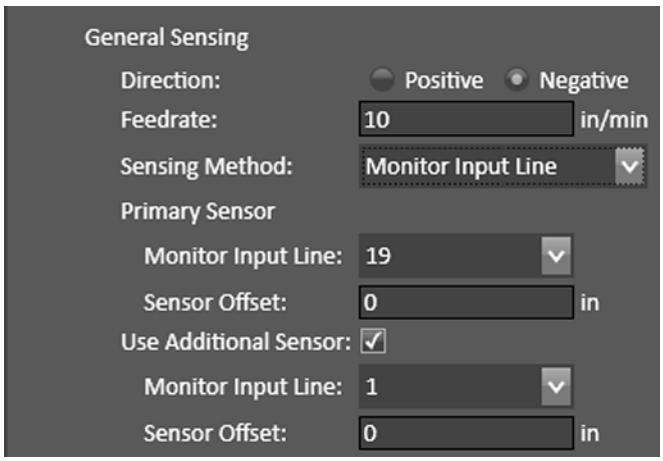
  Use Tool Sensing – Select to enable.

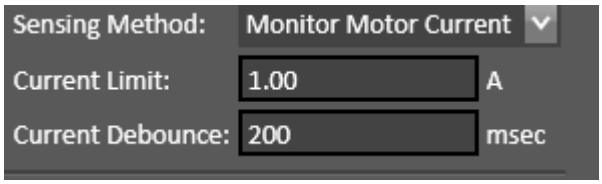
  Lift Axis Retract Position

  Lift Axis Sensing Start Position

  Direction – The direction of the move towards the sensor.

  Feed rate – The feed rate for the move towards the sensor.



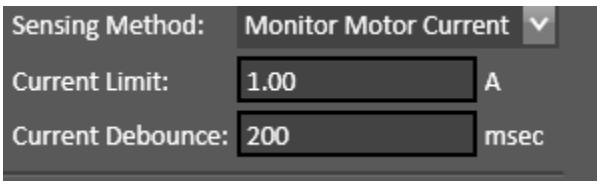


Tool Length Formula – The Z-Axis offset for the current tool is automatically entered into the Tool Library panel of the Configuration dialog box, according to the Tool Length Formula.

Chuck Sensing Offset

Check Sensing Offset Axis – Select axis, positive or negative.

- **General Sensing**



Direction – Select direction.

Feed rate – Enter value in in/min.

Sensing Method – Select either **Monitor Input Line** or **Monitor Input Current**.

Sensor Offset – The extra distance that the Z-Axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch or Z-Axis mechanical system.

Use Additional Sensor – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

If Monitor Input Current is selected, the following parameters apply:

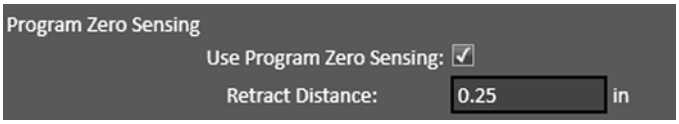
Current Limit – in amperes.

Current debounce – in msec.

- **Program Zero Sensing**

Use Program Zero Sensing – Select to enable.

Retract Distance – The amount that the Fab Head will move up after sensing the surface.



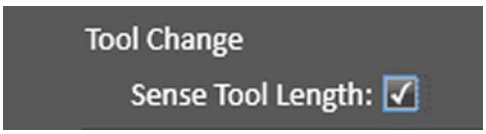
- **Tool Change**

Sense Tool Length – Select to enable.

- **Tool Changing**

Select **Silent, Manual,** or **Automatic.**

If **Automatic** is selected, the following parameters apply:

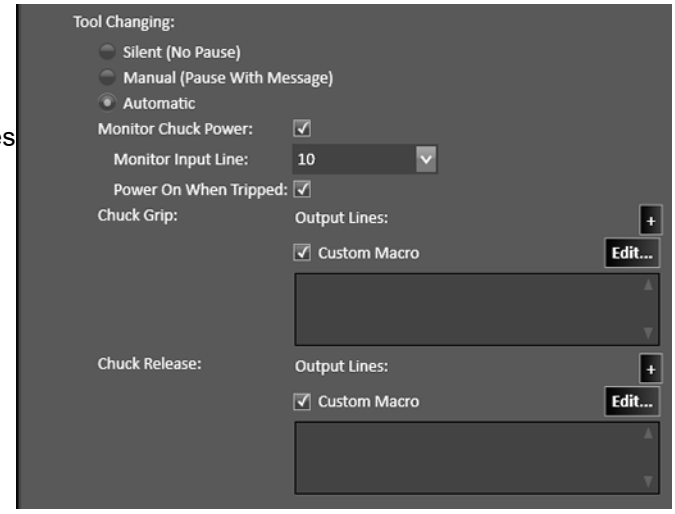


- **Monitor Chuck Power** – Select to enable.

Monitor Input Line – Select line.

Power On When Tripped – Select to enable.

Chuck Grip/Release Output Lines – When selected, define output line(s) used to control the Chuck. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.



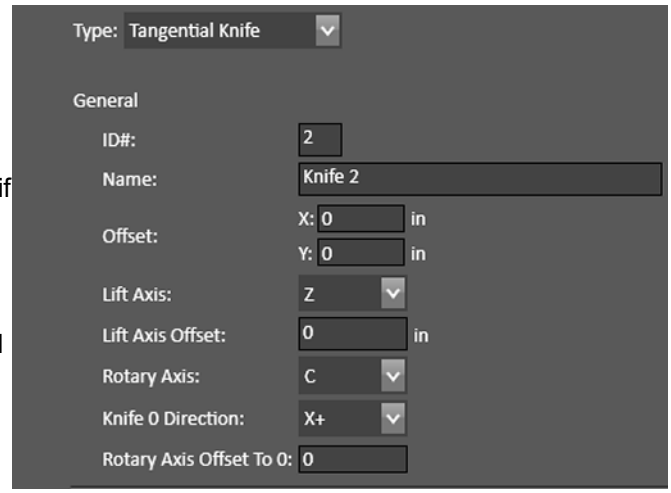
Custom Macro – Select these fields to define operations that take place when the Chuck Grip/Release is actuated. Selecting Edit will bring up the code editor window.

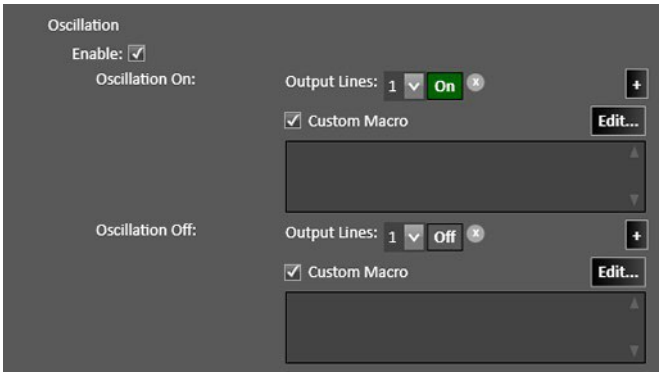
12.33 Tangential Knife

- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset –** Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis –** Select axis to be used when lifting the Fab Head.
- **Lift Axis Offset –** Enter value.
- **Rotary Axis** – Select axis.
- **Knife 0 Direction** – Select direction.

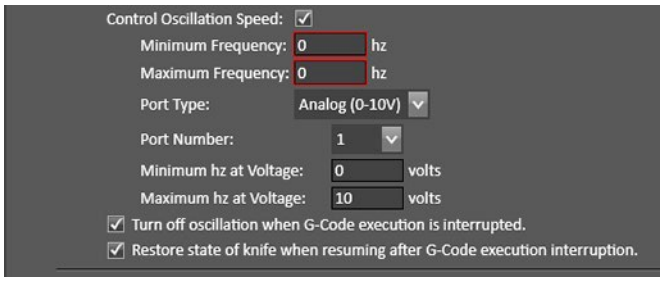
  Rotary Axis Offset To 0
- **Oscillation**

  Spindle Control – Select to enable.





Spindle Direction – Select Single- or Bi-Direction.

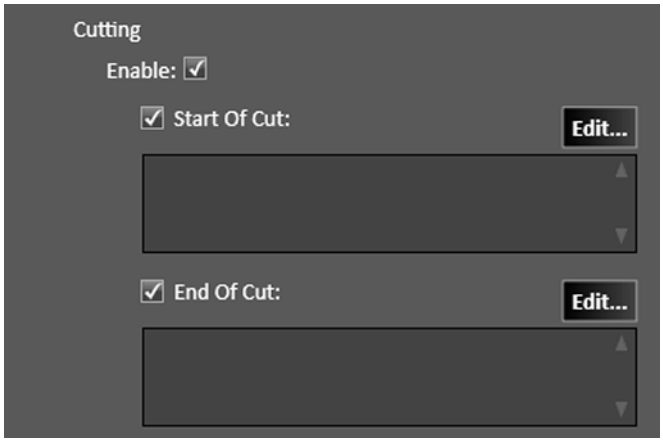


Spindle On/Off Output Lines – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

Control Oscillation Speed – Select to enable.

Minimum/Maximum Frequency – Enter value in hz.

Port Type – Options are Analog, Pulse Width Modulation, and Pulse Frequency Modulation.



Port Type – Select **Analog, Pulse-Width Modulation (PWM),** or **Pulse-Frequency Modulation (PFM).**

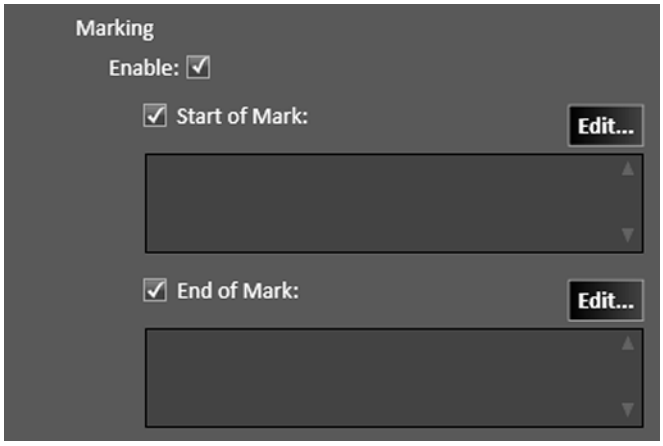
For **Analog**, select **Port Number**. For **PWM**, enter **Frequency.** For **PFM**, enter **Duty Cycle**.

Minimum/Maximum RPM at Voltage – Range varies for different port types.

G-Code execution interruption – Select to configure spindle behavior when G-Code is interrupted or when resuming.

- **Cutting**

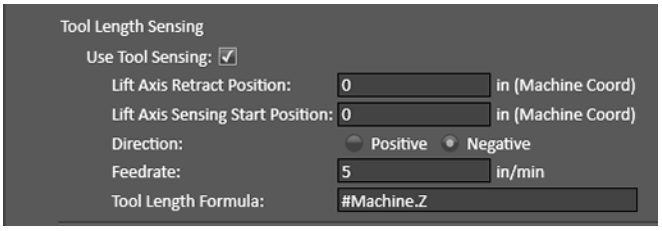
Start/End of Cut – When selected displays operations to be executed. Selecting **Edit** will bring up the code editor window.



- **Marking**

Start/End of Mark – When selected displays operations to be executed. Selecting Edit will bring up the code editor window.

- **Tool Length Sensing**



  Use Tool Sensing – Select to enable.

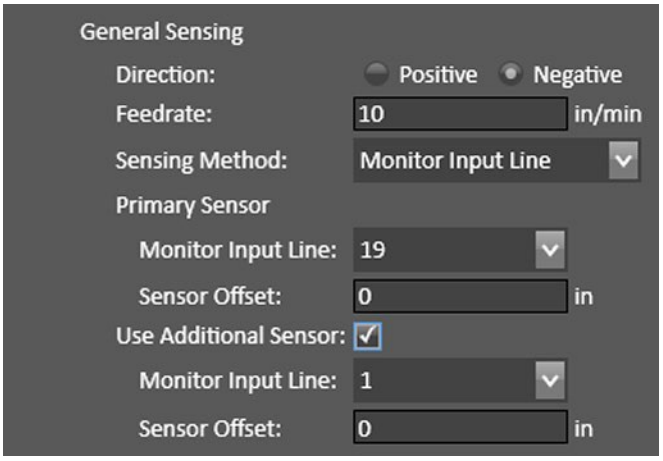
  Lift Axis Retract Position

  Lift Axis Sensing Start Position

  Direction – The direction of the move towards the sensor.

  Feed rate – The feed rate for the move towards the sensor.

- **General Sensing**



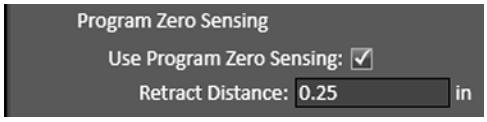
  Direction – Select direction.

  Feed rate – Enter value in in/min.

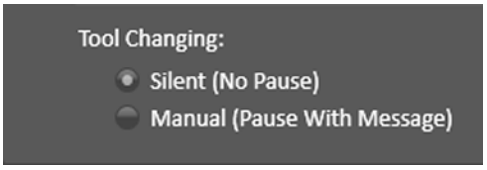
  Sensing Method – Select either **Monitor Input Line** or **Monitor Input Current.**

  Sensor Offset – The extra distance that the Z-Axis

- **Program Zero Sensing**

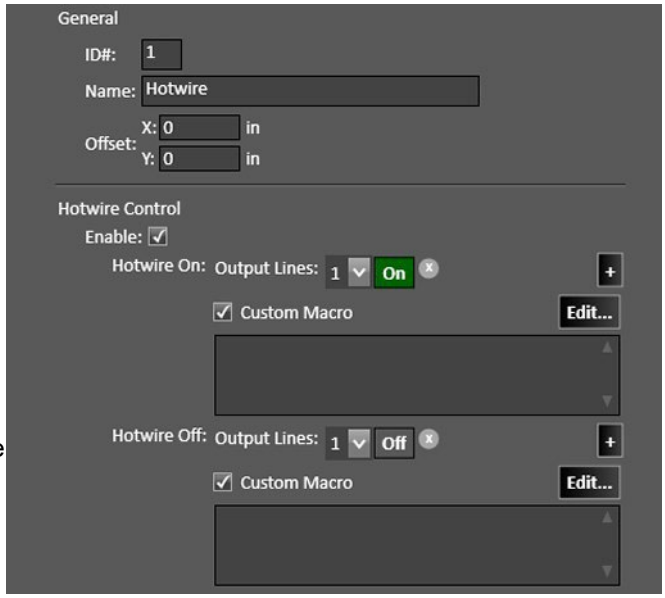


- **Tool Changing**



  Select **Silent** or **Manual**

12.34 Hotwire



- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Hotwire Control** – When selected, define output line(s) used to control the Fab Head. Add additional output lines

by selecting the + icon; subtract output lines by selecting the **x**.

12.35 Cut Charts

Cut charts are tables of information for configuring the properties of different manufacturer's Fab Heads. The information is stored in *.csv files in the FlashCut program files folder.

When configuring Fab Heads, the initial values for a particular head are populated from the relevant standard cut chart.

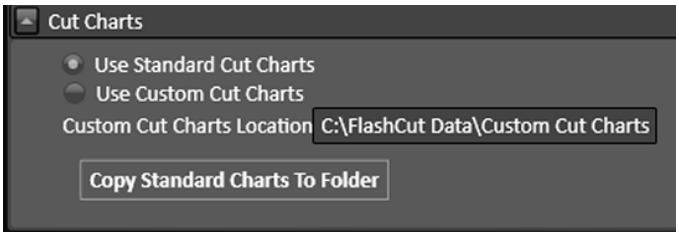
An example is shown below:

	NozzleMc	TorchLead	MaterialT	Thickness	KerfWidth	CutHeight	PierceHei	PierceDel	BestFeedr	BestVolta	Productio	Productio	Current	Pressure	EdgeStart	Units
2	125A Shie	25	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	85	N	English
3	125A Shie	50	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	85	N	English
4	125A Shie	75	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	95	N	English
5	125A Shie	25	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	85	N	English
6	125A Shie	50	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	85	N	English
7	125A Shie	75	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	95	N	English
8	125A Shie	25	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	85	N	English
9	125A Shie	50	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	85	N	English
10	125A Shie	75	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	95	N	English
11	125A Shie	25	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	85	N	English
12	125A Shie	50	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	85	N	English
13	125A Shie	75	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	95	N	English
14	125A Shie	25	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	85	N	English
15	125A Shie	50	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	85	N	English
16	125A Shie	75	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	95	N	English
17	125A Shie	25	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	85	N	English
18	125A Shie	50	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	85	N	English
19	125A Shie	75	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	95	N	English
20	125A Shie	25	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	85	N	English
21	125A Shie	50	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	85	N	English
22	125A Shie	75	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	95	N	English
23	125A Shie	25	Mild Steel	1.25	0.15	0.18	0.18	1	16	174	20	172	125	85	Y	English

Powermax125CutChart

12.35.1 Use Standard Cut Charts

Use the pre-loaded cut charts with recommended settings such as feed rate, pierce height, cut height and THC set point.



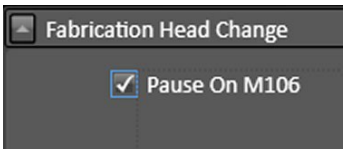
12.35.2 Use Custom Cut Charts

If you choose the Custom Cut Chart option, then a *.csv file copy of the standard cut charts will be stored into the specified location when you hit the **Copy Standard Charts To Folder** button.

Any value in these cut charts can be edited using a spreadsheet program like Excel.

NOTES:

13.0 Fabrication Head Change



13.1.1 Pause on M106

Select to enable.

13.1.2 Axes

- Motor Line/Axis Mapping
- Axis Settings
- Feed rates and Ramping
- Drive Parameters

13.1.3 Motor Line/Axis Mapping

FlashCut provides three (3) signals for motor drivers: step, direction, and enable. Different driver manufacturers have varying requirements for the polarity and timing of these signals. FlashCut provides the flexibility to tailor the motor signals to run any driver. Motor lines are the physical lines on the signal generator used for driving motors.

Each column is described below:

Line #

This number corresponds to the axis number on the back of the CNC controller. Each signal generator can command up to five (5) independent signals simultaneously.

Axis Letter

Assigns a logical axis letter (X, Y, Z, A/W, B/V, C/U) to a physical signal generator motor line (1, 2, 3, 4, 5). If you have more than one motor driving an axis on your machine, you may assign the same axis letter to more than one motor line.

NOTE

In general, axes X Y Z are always linear axes. Axes W V U could be linear or rotary depending on the specific machine.

Sub Axis

Index number that allows the rest of the FlashCut system to distinguish between multiple motors used on the same axis. This is important for homing, when FlashCut handles each motor independently.

Offset

When the system homes an axis that includes multiple sub axes and each sub axis has an independent switch, that total distance each sub axis retracts from its homing switch after making contact is:

Home switch offset for the axis + sub axis offset.

If one homing switch is used for multiple sub axes, or if multiple sub axes are bot being used, this sub axis offset value should be zero.

Direction

Sets the direction of rotation for the motor connected to each signal generator axis. Depending on how a motor is wired, the same signal from the motor driver can turn it clockwise or counterclockwise.

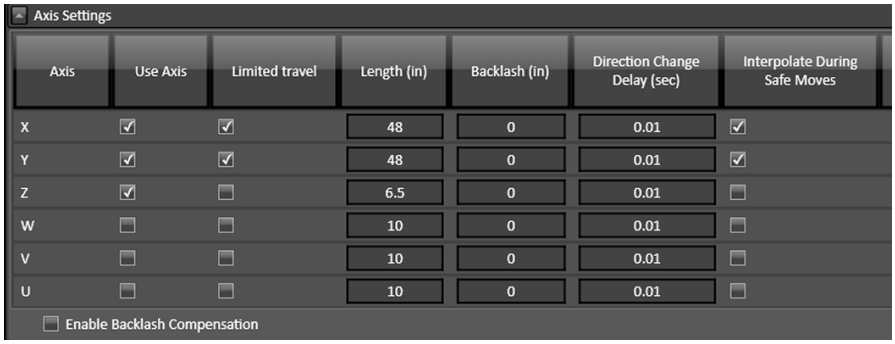
Use the jog buttons to make sure that a positive move in each axis on the screen corresponds to a positive move in each axis on the machine tool. Note that the direction of movement is defined as the direction of the tool relative to the workpiece. For example, on a typical milling machine, a positive X move (tool movement to the right) requires table movement to the left. If any direction is incorrect, change the Motor Direction from Positive to Negative (or vice-versa) to reverse the correspondence between the software and machine tool.

13.2 Axis Settings

Enables you to define whether or not an axis is being used and if there are physical limits to each of the axes in use.

13.2.1 Axis

These are the six possible axis identifiers that are expected to be found in a G-Code program. If the G- Code program calls any of these axis letters, the axis must be selected with the Use Axis checkbox. There are additional parameters that are specific to each axis.



13.2.2 Use Axis

Indicates whether or not the axis is used on your machine tool.

13.2.3 Limited Travel

Indicates whether or not the axis travel is limited. Many rotary axes do not have limited travel.

13.2.4 Length

Sets the length of travel in inches of each axis of your machine tool. You may want to define the axis length slightly smaller than the values published by the machine tool manufacturer. This will leave some room for error.

13.2.5 Backlash

Enter the backlash value for this axis here. See below for a more detailed explanation of backlash.

13.2.6 Direction Change Delay

Enter the desired direction change delay for each axis.

The direction change delay is a brief pause that occurs when a motor changes direction. It gives the stepper motor time to settle and come to a complete rest before moving in the opposite direction. A short delay helps prevent loss of position on some stepper systems, depending on the driver, motor and mechanics. A delay can also improve part quality in some applications. Note that the direction change delay is not used when a motor changes direction during circular interpolation or during a continuous contouring motion.

13.2.7 Interpolate During Safe Moves

FlashCut CNC does not always move all axes simultaneously. The sequence of axis motions follows a general-purpose scheme based on three fields in the Advanced Definition and Basic Homing Panels of the Configuration dialog box: Home End, Home Order, and Point Move Linear Interpolate. These fields should be configured to allow for safe moves in Point mode, which generally means the tool retracts from the workpiece before any other motion occurs. FlashCut uses the following rules to sequence the individual moves:

1. Do all non-interpolated moves that are towards the Home End, in the Home Order.
2. Do the interpolated move, if any.
3. Do all non-interpolated moves that are away from the Home End, in reverse Home Order.

By setting the three (3) configuration fields properly, you can make the sequence safe for your machine tool configuration.

13.2.8 Enable Backlash Compensation

When checked, FlashCut will use backlash compensation for all direction changes.

13.2.9 On Backlash Compensation

Backlash Compensation: When an axis changes direction, a small turn of the motor to move the drive mechanism through any mechanical play, before resuming motion.

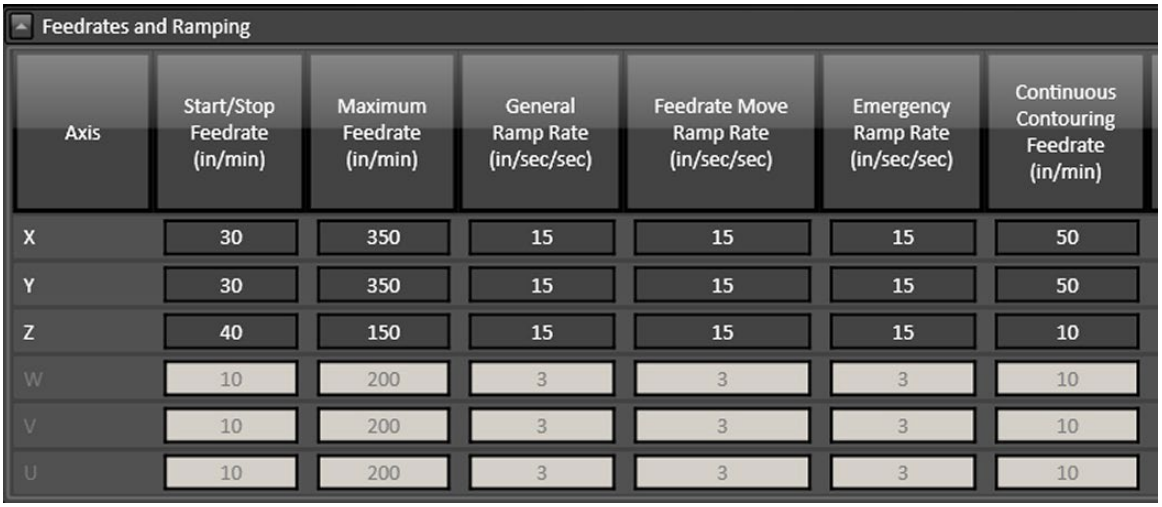
CNC control works best if there is little or no backlash on the machine tool. However, if your machine doesn't have a zero-backlash drive mechanism (such as ball screws or

anti-backlash nuts), FlashCut can compensate for the backlash. Immediately after an axis changes direction, FlashCut turns the motor a small amount to absorb the backlash before performing the next commanded move.

13.2.10 To Set Backlash

1. Make sure the Enable Backlash Compensation checkbox is unchecked.
2. In CNC, choose the Jog panel.
3. Drive the X axis in either direction at least 0.25" (to take out any backlash in that direction).
4. Zero the relative coordinates.
5. Choose the Step option. Jog the axis step by step in the opposite direction until you detect table movement (using a dial indicator).
6. The Relative coordinate X axis value is the amount of X axis backlash on your machine tool.
7. Write down this number and repeat the above process at different places along the X axis.
8. Record the average of all backlash values in the X axis Backlash text box. If you have no backlash on an axis, or if you don't want backlash compensation on an axis, enter zero.
9. Repeat the above steps for each axis.
10. When finished, check the Enable Backlash Compensation checkbox.

13.3 Feed Rates and Ramping


Feedrates and Ramping

Axis	Start/Stop Feedrate (in/min)	Maximum Feedrate (in/min)	General Ramp Rate (in/sec/sec)	Feedrate Move Ramp Rate (in/sec/sec)	Emergency Ramp Rate (in/sec/sec)	Continuous Contouring Feedrate (in/min)
X	30	350	15	15	15	50
Y	30	350	15	15	15	50
Z	40	150	15	15	15	10
W	10	200	3	3	3	10
V	10	200	3	3	3	10
U	10	200	3	3	3	10

For your system to run accurately, it's important to tune the values on this panel correctly for your specific combination of drivers, motors and machine tool. All parameters must be set conservatively to prevent the system from losing position (losing steps).

An axis can lose a step when the motor gets the signal to move a step but isn't physically able to perform the move. The usual cause is insufficient torque at a given motor RPM. Since most stepper motors are open loop systems, there is no way of telling when a step is lost without physically measuring the movement of the axis and comparing that to the amount it should have moved. However, when stepper motors are not over torqued they are very reliable and accurate.

For that reason, we highly recommend finding the values for the following settings at which steps are not lost, then limiting the settings to about 70% of those values. Due to variations in the drive mechanism for each axis, make sure you follow the instructions below at several positions along each axis, in both directions:

13.3.1 Axis

These are the six (6) possible axis identifiers that are expected to be found in a G-Code program. If the G- Code program calls any of these axis letters, the axis must be selected with the Use Axis checkbox. There are additional parameters that are specific to each axis.

13.3.2 Start/Stop Feed Rate (in/min)

Sets the initial and ending feed rate for ramped moves. To set the start/stop feed rates:

1. Enter 499 for the X-Axis Start-Stop Feedrate and 500 for the Maximum Feedrate and then choose OK.

   If your software settings do not allow you to set the Maximum Feedrate to 500, set it to the largest value the software allows and set the Start/Stop feedrate to 1 less than that.

2. All settings in these steps assume inch units; if your machine is set for metric units, you should multiply these settings by 25.

3. Choose the Point button in the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Feedrate from the Rate Mode pull-down menu and enter a relatively slow feedrate (such as 5) in the

Linear Feedrate text box. Make sure you have room to move the X axis 1 inch, and then choose Move.

4. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.
5. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.
6. Now run the entire length of the X axis in both directions to make sure there is no slippage at any point on the entire axis.
7. Choose Feedrate/Ramping from the Configuration menu.
8. Enter 70% of the value you found in the Start/Stop Feedrate text box for the X axis, and then choose OK.
9. Repeat this process for all axes.

13.3.3 Maximum Feedrate (in/min)

Sets the maximum feedrate for the system. No matter how fast the system is commanded to move (from the G-Code, Point, Jog or MDI control panels), the axis will never move faster than this value. The system will use ramping (acceleration and deceleration) to achieve this feedrate.

To set the maximum feedrates:

Make sure you've set the Start/Stop Feedrates correctly as described above.

1. Enter 1 in/sec2 in the General Ramping Rate text box for the X axis. (This is a conservative ramping rate.)
2. Set the Maximum Feedrate to 500 in/min (or the highest value your settings allow), and then choose OK.
3. Choose the Point button on the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Feedrate from the Rate Mode pull-down menu and enter a feedrate that is double the Start/Stop Feedrate for the X axis. Make sure you have room to move the X axis 1 inch, and then choose Move.
4. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't

slip, try a faster feedrate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.

5. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.
6. Now run the entire length of the X axis in both directions to make sure there is no slippage at any point on the entire axis.
7. Choose Feedrate/Ramping from the Configuration menu.
8. Enter 70% of the highest no-slip feedrate you found in the X axis Maximum Feedrate text box.
9. Repeat this process for all axes.

13.3.4 General Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for rapid moves and jogging.

Ramping Rates typically range from 1 to 10 in/sec^2. Slower ramping rates require more time to ramp up to the maximum feedrate and to ramp down to a stop. This may become a potentially dangerous situation when using the Feed Hold button or jogging since the machine will take longer to come to a complete stop. The goal is to choose a fast-ramping rate that will start and stop the tool responsively without losing steps. Fast ramping rates can also allow acceleration past resonant speeds of a stepper motor.

Make sure you've set the Start/Stop and Maximum Feedrates correctly as described above.

1. Enter 1 in/sec2 in the General Ramping Rate text box for the X axis. (This is a conservative ramping rate.)
2. Choose OK.
3. Choose the Point button on the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Rapid from the Rate Mode pull-down menu. Make sure you have room to move the X axis 1 inch, and then choose Move. If the table doesn't reach a continuous speed before ramping down to a stop, there isn't enough room to fully ramp up and down, and you should increase the length of the move.
4. If the motor slips, repeat this process with a lower ramping rate. If the motor doesn't slip, try a higher ramping rate. Note that slight slippage can be detected

using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.

5. Repeat the above steps until you determine an optimal General Ramping Rate for the X axis.
6. Choose Feedrate/Ramping from the Configuration menu.

13.3.5 Feedrate Move Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for feedrate moves.

13.3.6 Emergency Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration used in emergency operations, e.g. Feedrate Hold (Emergency).

13.3.7 Continuous Contouring Feedrate (in/sec)

Continuous contouring (also called Look-ahead) is a high-performance feature that lets FlashCut run G-Code files smoother and faster. The advantages of using continuous contouring are:

- Smooth transitions from one G-Code line to the next if the two moves are in line with one another. There is no ramp-down period and subsequent ramp up period. This minimizes witness marks, burning, melting and other quality problems sometimes encountered in machining. It also provides smooth, high-speed motion for any other control application that requires it. A continuous smooth motion may span hundreds or thousands of G-Code lines.
- High speed machining of point-to-point type G-Code files (composed of many short moves) used in complex 2D profiling and 3D surfacing. These G-Codes files can be executed much faster with continuous contouring.

To determine if two moves are lined up enough to allow a smooth transition without ramping, FlashCut calculates the change in speed of each motor through the transition and compares that to the setting for continuous contouring feedrate tolerance. If the change in speed for each motor is less than the tolerance, FlashCut executes the two moves as part of the same smooth motion. Otherwise, FlashCut ramps down to end one motion and ramps up to begin another.

Any command that breaks the continuity of motion causes the machine tool to ramp

down to a stop (such as any M code to control an auxiliary device, a G04 command for dwell, and so on).

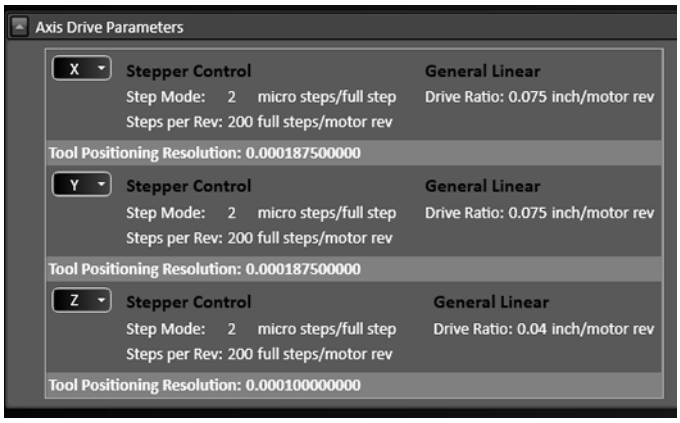
A typical default setting for the continuous contouring feedrate tolerance is the **Start/Stop Feedrate (in/min)** value. The higher you set the tolerance, the smoother a G-Code file will be executed (i.e., more moves will be linked together into smooth motions). If the values are set too high, loss of steps (stepper motor drivers) or servo errors (digital servo drivers) can occur. The best way to find an optimal setting is through trial and error. Incrementally raise the values and run demanding test files, checking for lost steps or servo errors.

Continuous contouring is always turned on. If you prefer that all moves ramp up and down, set the Continuous Contouring Feedrate Tolerance to zero for all axes.

13.4 Drive Parameters

13.4.1 Axis Drive Parameters

The axis drive parameters window displays the following operation:

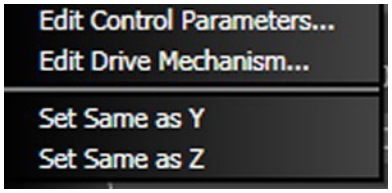


For each axis:

- The current Control Parameters and Drive Mechanism are displayed.
- The Tool Positioning Resolution is displayed. This is the length of axis movement per Signal Generator pulse, automatically calculated from the control and drive mechanism settings.

This value represents the smallest linear increment that the machine can move on each particular axis. More specifically it is the exact linear distance traveled by the machine when the motor rotates by the smallest possible increment, also known as a step.

- A dropdown menu displays the following options (X-Axis shown):



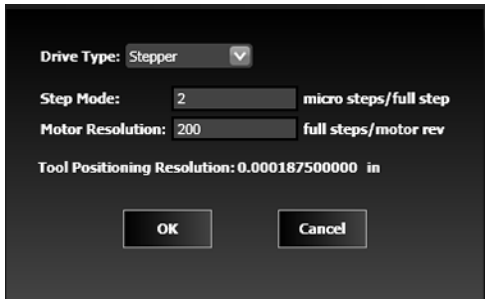
1. **Set Same as**

   Sets all control/drive mechanism settings the same as another axis.

2. **Edit Control Parameters**

   Opens the Control Parameters dialog box. Select Stepper or Servo from the Driver Type pulldown menu.

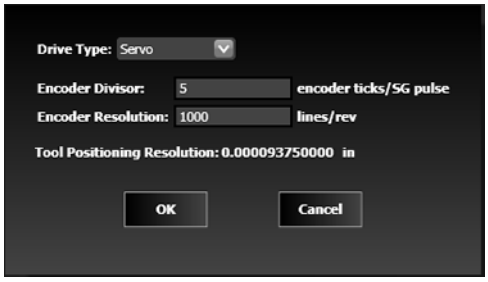
   - **Stepper**



   Step Mode – The number of micro steps between each full motor step. Enter "1" for full-stepping, "2" for half-stepping, "4" for quarter-stepping and so on.

   Motor Resolution - The number of full motor steps for one revolution of the motor. For example, a 1.8º stepper motor has 200 full steps per revolution; a 0.9º stepper motor has 400 full steps per revolution.

   - **Servo**



   Encoder Divisor – The number of encoder pulses that correspond to one pulse from the Signal Generator.

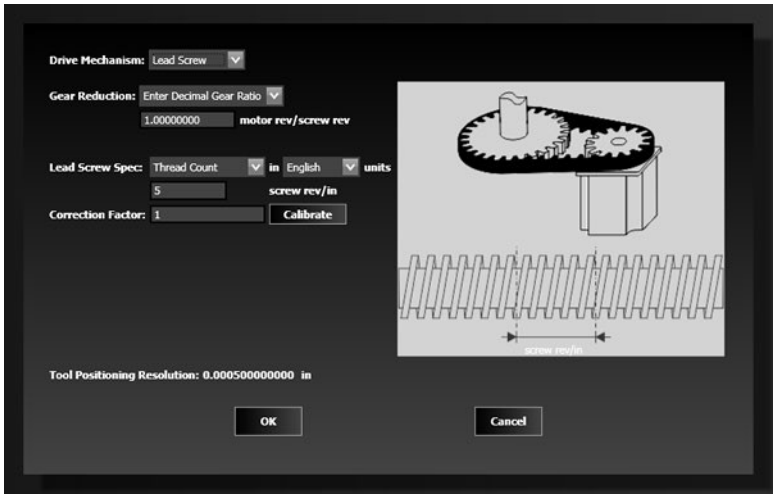
   Encoder Resolution – The number of encoder pulses per revolution, not including any adjustment for running in quadrature mode. When calculating the Tool Positioning Resolution, FlashCut assumes the encoder is running in quadrature mode.

13.5 Edit Drive Mechanism

13.5.1 Edit Control Parameters

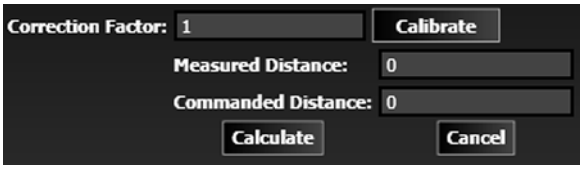
Opens the Control Parameters dialog box. Select Stepper or Servo from the Driver Type pulldown menu.

- **Lead Screw**



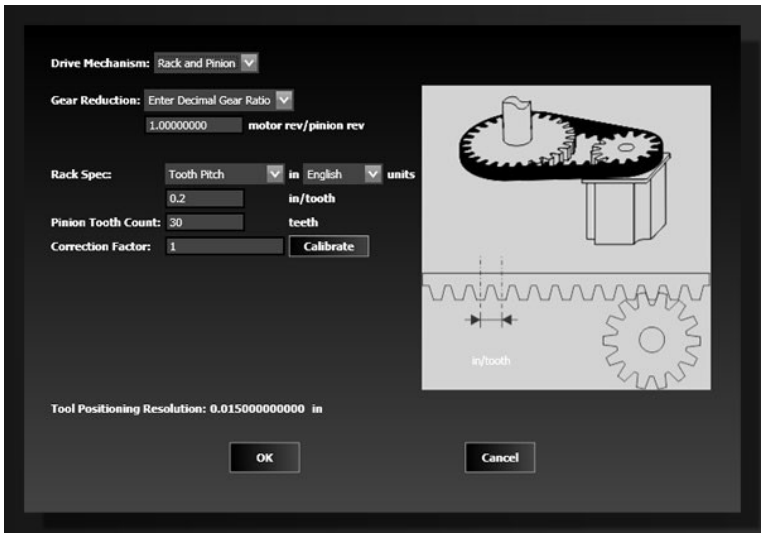
Gear Reduction – Enter the parameters for any gearing between the motor and lead screw. A decimal gear ration of 1.0 indicates no gearing.

Lead Screw Specification – Enter the parameters for the lead screw.

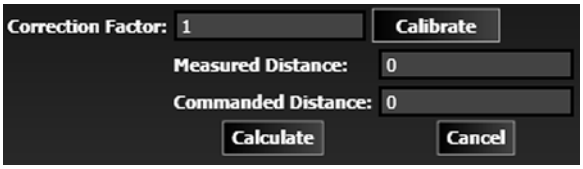


Correction Factor – The operator commands the machine to move a specific distance and then measures the actual distance travelled. The system then calculates the correction factor for the tool positioning resolution. Select Calibrate to display the controls.

- **Rack and Pinion**



Gear Reduction – Enter the parameters for any gearing between the motor and pinion. A decimal gear ratio of 1.0 indicates no gearing.

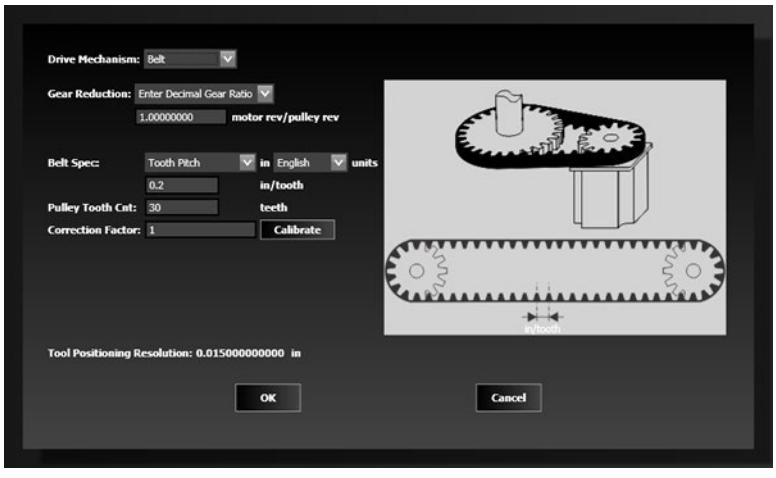


Rack Specification – Enter the parameters for the rack.

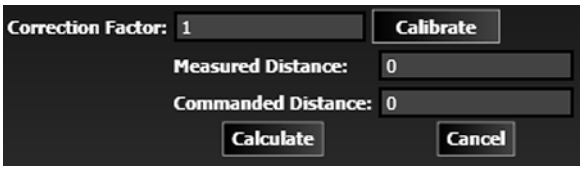
Pinion Tooth Count – The number of teeth on the pinion.

Correction Factor – The operator commands the machine to move a specific distance and then measures the actual distance travelled. The system then calculates the correction factor for the tool positioning resolution. Select Calibrate to display the controls.

- **Belt**



Gear Reduction – Enter the parameters for any gearing between the motor and drive pulley. A decimal gear ration of 1.0 indicates no gearing.

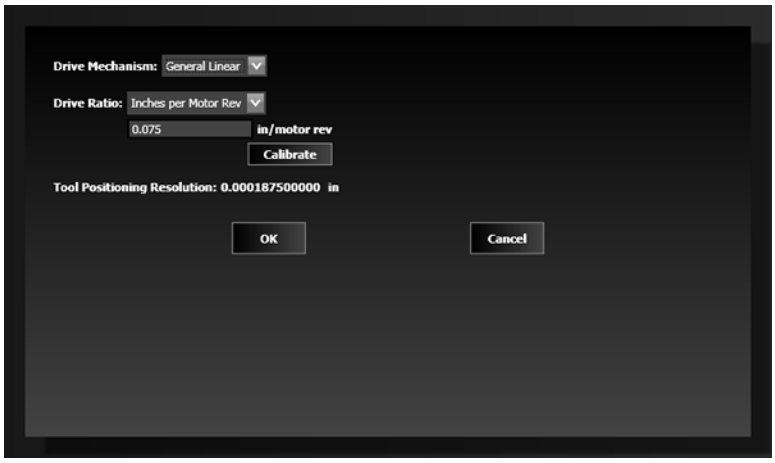


Belt Specification – Enter the parameters for the belt.

Pulley Tooth Count – The number of teeth on each pulley.

Correction Factor – The operator commands the machine to move a specific distance and then measures the actual distance travelled. The system then calculates the correction factor for the tool positioning resolution. Select Calibrate to display the controls.

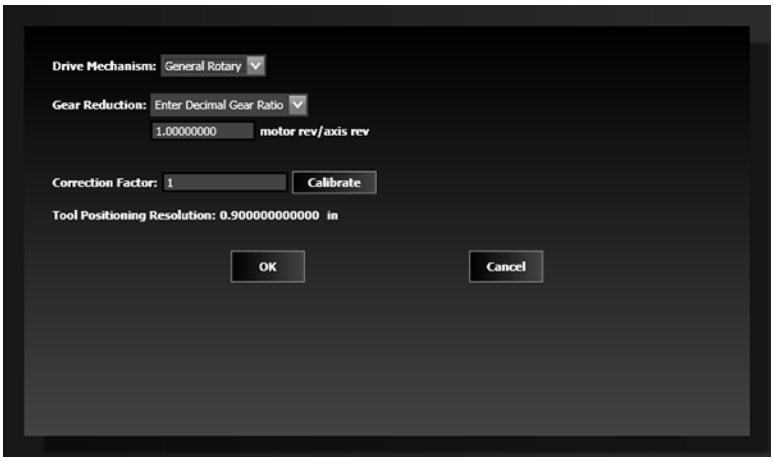
- General Linear



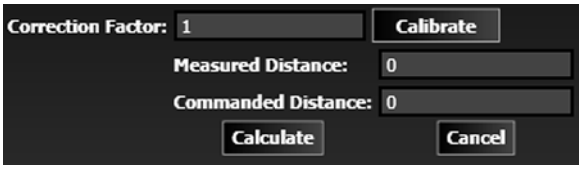
Drive Ratio – Enter the distance moved per motor revolution or the number of motor revolutions per unit distance.

Calibrate – Display the Measured Distance and Commanded Distance fields. The operator commands the machine to move a specific distance, and then measures the actual distance travelled. The system then calculated the correction factor for the tool positioning resolution.

- **General Rotary**



Gear Reduction – Enter the parameters for any gearing between the motor and the rotational axis. A decimal gear ration of 1.0 indicates no gearing.



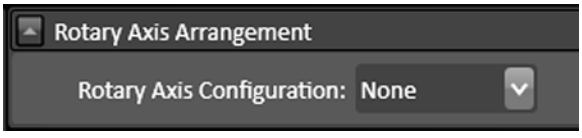
Correction Factor – The operator commands the machine to move a specific distance and then measures the actual distance travelled. The system then calculates the correction factor for the tool positioning resolution. Select Calibrate to display the controls.

13.6 Rotary Axes

If the system is using a rotary axis, it must be configured here. Define the type of rotary axis and how the rotary axis is positioned relative to the other axes. Additionally, you need to specify if the axis velocity will be commanded by a linear or rotary feedrate.

13.6.1 Rotary Axes Arrangement

Click the arrow next to Rotary Axis Configuration to display a dropdown list box.



You may choose from the following configuration options:

1. **None**

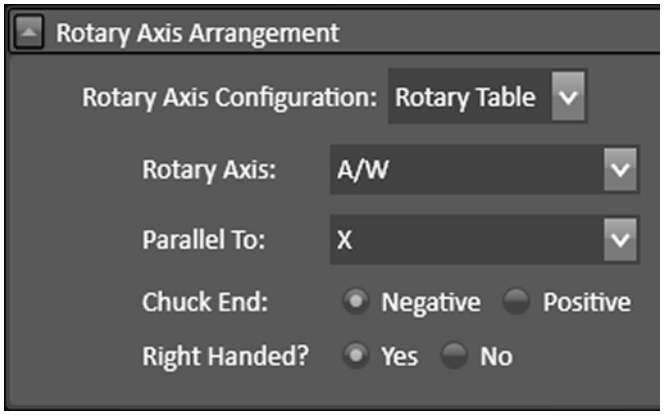
Select None if you are not using a rotary axis.

2. **Other**

   Select Other if you are using some other configuration.

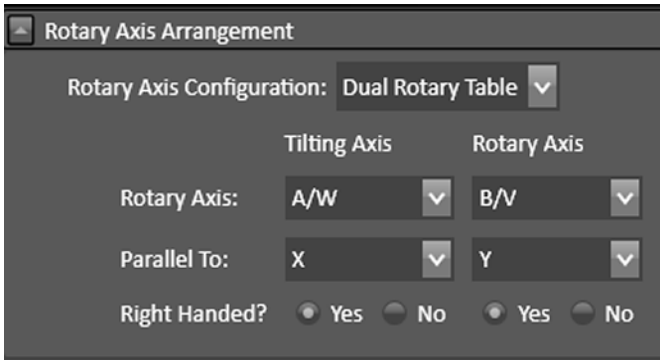
3. **Rotary Table**

   Select Rotary Table if you have only one rotary axis. Righthanded indicates whether or not the rotary axis obeys the righthand rule. If you curl you hand closed in the direction the rotary axis revolves and your thumb points in the positive direction of the axis of rotation (Parallel Axis setting), the axis is righthanded.



4. **Dual Rotary Table**

   Select Dual Rotary Table if you are using two (2) rotary axes built into a single table. You must define a Rotary Axis for each.



These options are part of the rotary table configuration screens.

- Use offset for rotary zero.

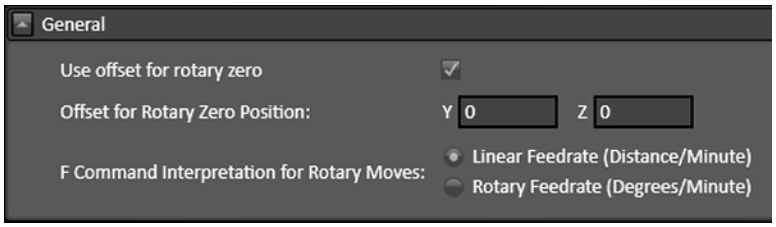
  If checked, define the offset below.

- Offset for Rotary Zero Position

  Indicates the fixture offset for the rotary zero position.

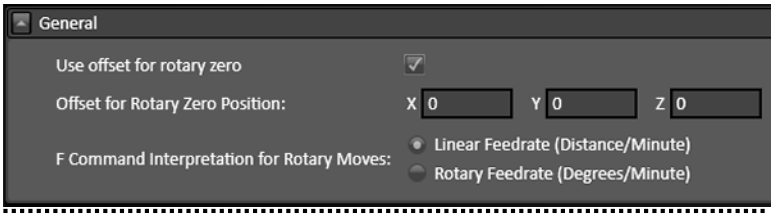
  - 4-Axis Machining

    Rotary zero is anywhere along the axis of rotation of the rotary axis. The parallel axis in the fixture offset is ignored.



  - 5-Axis Machining

    Rotary zero is at the intersection of the axes of rotation of both rotary axes.



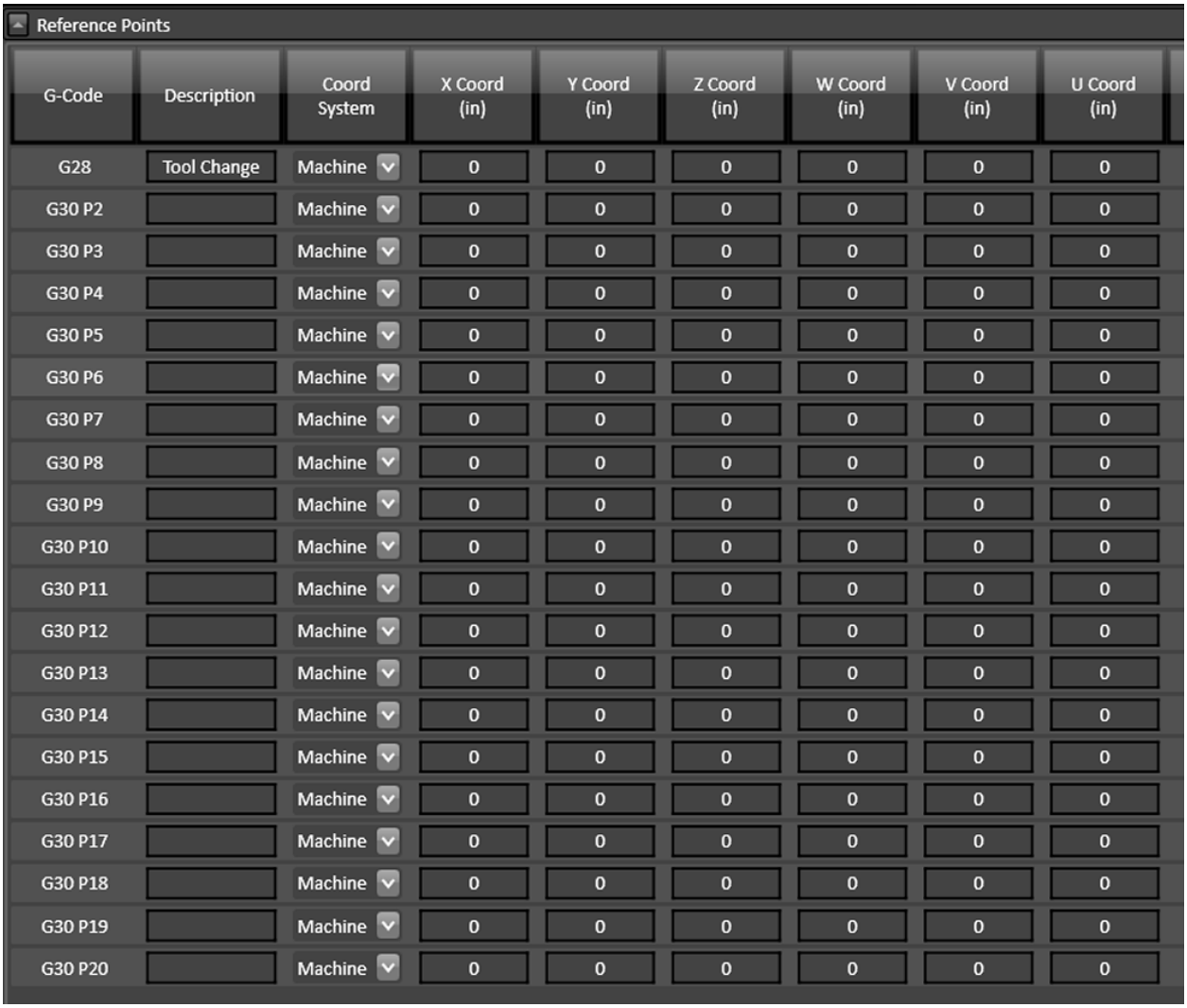
- F Command Interpretation for Rotary Moves

  For moves that include rotary motion, this setting tells FlashCut whether to interpret the feedrate command (F) as a linear (distance/minute) or rotary (degrees/minute) feedrate.

13.7 Reference Points

13.7.1 Reference Points

Reference points provide a way to move the machine tool to a predefined location. They are accessible using the G28 and G30 commands, described in the Programming Reference.



G-Code	Description	Coord System	X Coord (in)	Y Coord (in)	Z Coord (in)	W Coord (in)	V Coord (in)	U Coord (in)
G28	Tool Change	Machine	0	0	0	0	0	0
G30 P2		Machine	0	0	0	0	0	0
G30 P3		Machine	0	0	0	0	0	0
G30 P4		Machine	0	0	0	0	0	0
G30 P5		Machine	0	0	0	0	0	0
G30 P6		Machine	0	0	0	0	0	0
G30 P7		Machine	0	0	0	0	0	0
G30 P8		Machine	0	0	0	0	0	0
G30 P9		Machine	0	0	0	0	0	0
G30 P10		Machine	0	0	0	0	0	0
G30 P11		Machine	0	0	0	0	0	0
G30 P12		Machine	0	0	0	0	0	0
G30 P13		Machine	0	0	0	0	0	0
G30 P14		Machine	0	0	0	0	0	0
G30 P15		Machine	0	0	0	0	0	0
G30 P16		Machine	0	0	0	0	0	0
G30 P17		Machine	0	0	0	0	0	0
G30 P18		Machine	0	0	0	0	0	0
G30 P19		Machine	0	0	0	0	0	0
G30 P20		Machine	0	0	0	0	0	0

1. **G-Code**

   This field identifies the command in a G-Code file that moves the machine to the point (not editable).

2. **Description**

   Enter a description for the point. For G28, the standard description is "Tool Change".

3. **Coord System**

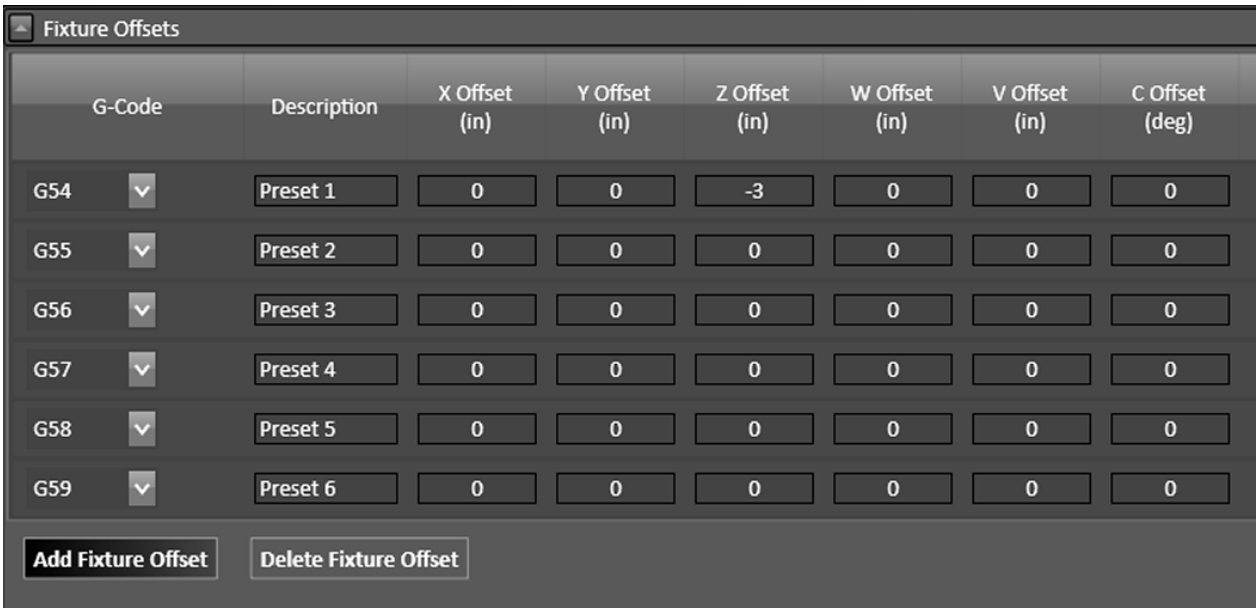
Choose between machine and program coordinates.

4. **X. Y, Z, W, V, U Coordinantes**
   Enter the coordinate for each axis.

13.8 Fixture Offsets

13.8.1 Fixture Offsets

Fixture offsets provide a way to set Program Zero to a predefined location. They are activated using the G54-G59 and G54.1 commands (see Programming Reference). The XYZWVC coordinates are defined in machine coordinates. These coordinates represent the offset from Machine Zero to the new Program Zero location.



1. **G-Code**
   The field identifies the 'G' command in a G-Code file that sets Program Zero the offset (not editable).
2. **Description**
   Enter a description for the offset.
3. **X, Y, Z, W, V, C Offset**

   For each axis, enter the offset from machine zero to program zero (i.e., the machine coordinate the machine tool is at the desired program zero location).

13.9 Homing

13.9.1 Axis Settings

For each axis, the following settings are available:

1. **Home Axis**

   Select to configure homing for the axis.

2. **Home End**

   The end of the axis at Machine Zero (Home).

3. **Home Order**

   The axis order for the homing operation. For safety, set the order so the initial homing move retracts the cutting tool away from the workpiece.

4. **Home Switch Offset**

   The distance each axis backs away from the home switch after the switch is tripped during homing.

5. **Homing Rate**

   The feedrate at which the machine tool will move when locating home switches.

6. **Homing Tolerance**

   The distance allowed from the original Machine Zero location to the new Machine Zero location found during homing.

7. **Execute M-Codes Before/After**

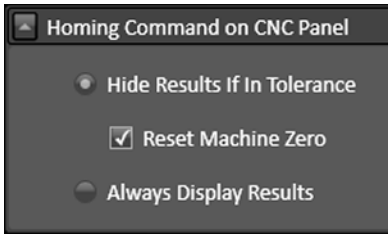
   You may specify M-Codes to execute before and after each axis homes. For more information on defining M-Codes see **M-Code Definitions**.

8. **Home Position Machine Coordinate**

If the home switch is positioned at the travel limit, the machine coordinate of the home switch is zero. If the home switch is positioned somewhere within the machine envelope, set this value to the machine coordinate of the home switch location (nonzero value).

13.10 Homing Command on CNC Panel

These settings apply to the homing process initiated by the Homing Command on the CNC Panel.



1. **Hide Results if in Tolerance**
   Tells FlashCut to display the results of re-homing if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

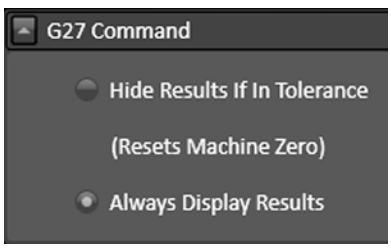
2. **Reset Machine Zero**
   Tells FlashCut to automatically reset Machine Zero to the new location found, if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

3. **Always Display Results**

   Tells FlashCut to always display the results of re-homing, even if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

13.11 G27 Command

These settings apply to the homing process initiated by the G27 Command. They work the same as the Seek Home Button controls described above, except that if you choose Hide Results if in Tolerance, FlashCut always resets Machine Zero to the new location found.



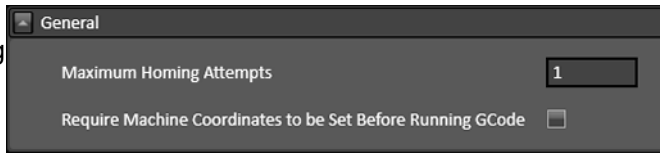
1. **Hide Results if in Tolerance**
   Tells FlashCut not to display the results of re-homing, if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

2. **Always Display Results**

Tells FlashCut to always display the results of re-homing, even if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

13.12 General

1. Maximum Homing Attempts
   The maximum number of times the machine tool will re-home, while trying to get the discrepancy from the previous Machine Zero location within the Homing Tolerance for all axes.



2. Require Machine Coordinates to be Set before Running G-Code

   When checked, machine coordinates must be set before the system can run a G-Code program.

13.13 Dynamic Ventilation

Dynamic ventilation systems sense the location of the cutting head, and only activate the ventilation system in the specific zone where it is needed. The ventilation system is distributed over several zones, each with a damper that can be opened and closed on command.

If your machine is equipped with a dynamic ventilation system, its behavior can be controlled through these settings. For example, by configuring the length of the overlap between adjacent zones, you can ensure that the dampers controlling the airflow to the ventilation system have sufficient time to open and close as the cutting head passes between zones.

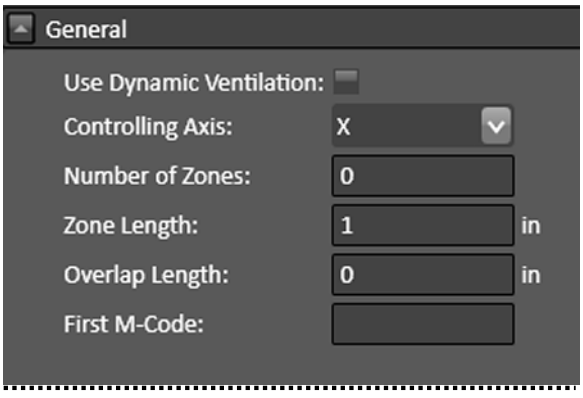
13.13.1 General

1. **Use Dynamic Ventilation**

   When checked, the dynamic ventilation system will be active.

2. **Controlling Axis**

   Sets the axis along which the ventilation zones are defined.



3. **Number of Zones**

   Sets the number of zones.

4. **Zone Length**

   Sets the length of each zone along the controlling axis.

5. **Overlap Length**

   Sets the amount of overlap between zones. Overlapping zones ensure that the dampers controlling the airflow have sufficient time to open and close.

6. **First M-Code**

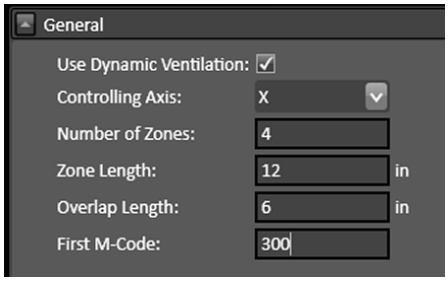
   Enter the first M-Code here. The first M-Code is the code that opens the damper in zone one. Note that codes are consecutive: the second M-Code will close the damper for zone one and so on.

   The first M-Code cannot use the numbers 0, 1, 2, 30, 98, 100, 101, 102, 103. The first M-Code must be in the list of defined M-Codes.

13.14 Dynamic Ventilation Example

The following steps illustrate how a dynamic ventilation system for a plasma table could be configured in FlashCut. The example assumes a table with four (4) 1-foot zones along the X-Axis, with a recommended overlap of six (6) inches. The values chosen are arbitrary and are only an example. Consult the manufacturer of your system to determine the optimal values for your ventilation system.

1. Enter the following values in the Dynamic Ventilation General configuration panel.



2. Define Output Line M-Codes 300, 301, 302, and 303 to control the 4 dampers in each zone on the X-Axis (see M-Code Definitions, below). It is important to use consecutive numbers for the codes and to begin with the zone starting at the X-Axis zero point.

3. Save your setup file.

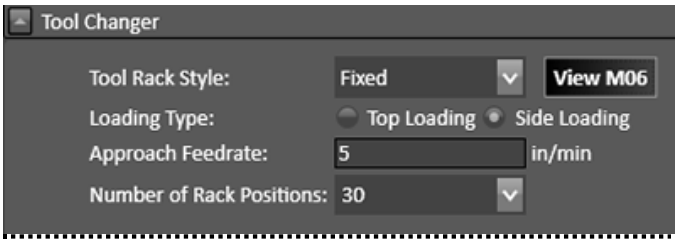
When the torch starts, the dynamic ventilation system will be active, and the damper in the first zone will be open. When the torch moves to a position 6 inches or less from the boundary of the adjacent zone, the damper in the second zone will open, and that zone will also be active. As the torch moves from left to right, the new damper to the right will open, and the old damper to the left will close. When the torch moves from right to left, the reverse will be true.

Again, consult the manufacturer of your plasma table and ventilation system to determine the optimal values for configuring your system.

13.15Tool Changer

FlashCut provides a great deal of flexibility in how it handles a tool change. It can accommodate any manual or automatic tool change scheme. For more configuration parameters, see Fab Head settings for **Mill.**

13.15.1 Tool Changer



1. **Tool Rack Style**

   Select Fixed, Carousel or Custom

2. **Loading Type**

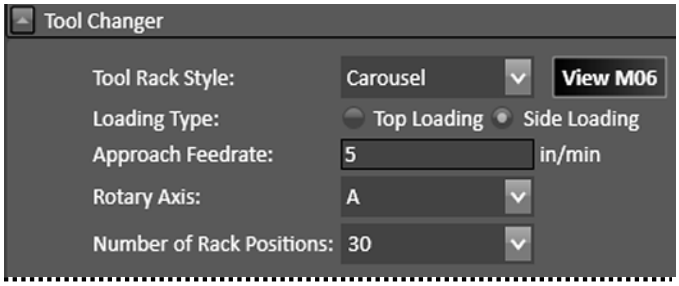
   Select Top or Side Loading

3. **Approach Feedrate**

   Feedrate for all moves to the Lift Axis Grip and Release positions.

4. **Rotary Axis**

   Select axis for Carousel/Custom types.



5. **Number of Rack Positions**

   Numbers refers to Tool Library entries.

6. **View M06**

   FlashCut executes the macro defined for M06 on the M-Code Definitions panel. The macro contains the logic required to drive an automatic tool changer. You can create a macro to control any automatic tool changer. Also, you can use the

macro to perform a customized sequence of operations for a manual tool change. Selecting View M06 brings up the macro editor.

7. **Tool Rack Positions**

Set the following parameters for each tool rack position in machine coordinates.

8. **X/Y Axis Positions**

   The machine coordinates of the tool rack position.

9. **Rotary Axis Positions**

   Carousel/Custom racks only.

10. **Loading Safe Positions X/Y**

    The machine coordinates of the X/Y loading positions. Side Loading racks only.

11. **Lift Axis Grip**

    Lift Axis machine coordinates where the tool chuck closes to grip the tool.

12. **Lift Axis Release**

    Lift Axis machine coordinates where the tool chuck opens to release the tool.

13. **Lift Axis Near Grip**

    When putting away a tool, the Z machine coordinates to which the machine moves all the rapid rate.

⚠ WARNING

It is very important to set this value high enough to accommodate the longest tool. Otherwise, a crash will occur. The safest value is zero (Z-Axis completely raised).

14. **Lift Axis Near Release**

When putting away a tool, the Lift Axis machine coordinates to which the machine moves at the rapid rate.

⚠ WARNING

It is very important to set this value high enough to accommodate the longest tool. Otherwise, a crash will occur. The safest value is zero (Z-Axis completely raised).

15. **Lift Axis Raised**

Lift Axis machine coordinate that's high enough to allow the longest tool to clear the tool rack and any workpiece or fixture when the machine moves to or from the tool rack position.

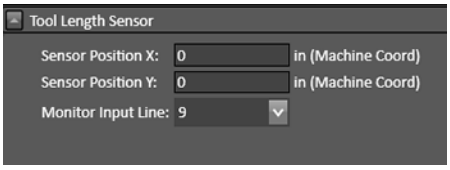
⚠ WARNING

It is very important to set this value high enough to accommodate the longest tool. Otherwise, a crash will occur.

13.16 Sensing

13.16.1 Tool Length Sensor

This panel sets parameters used in tool length sensing. Sensor Position X and Sensor Position Y must be specified in machine coordinates. For more parameters, see Feb Head settings for **Mill**.



13.17 Programming

- M-Code Definitions
- M-Code Execution
- G-Code

13.18 M-Code Definitions

The M-Code definition option presents these M-Code definition configuration types:

- Output Line M-Codes
- Macro M-Codes

13.19 Output Line M-Codes

FlashCut controls output lines to activate external devices such as a plasma torch or plate marking device, the M-Codes below are tied to output lines.

1. **M-Code**

   The number of the M-Code that you want to believe.

2. **Description**

   A brief description of the action taken when this M-Code is executed.

3. **Execute Before Move**

   Indicates the sequence of actions when there is a machine tool move command on the same program line as the M-Code. Check this box to execute the M-Code before the machine moves as commanded.

4. **Execute Delay First**

   Check this box to implement the specified delay before the M-Code. If the box remains unchecked the delay will occur immediately after the M-Code.

5. **Delay (sec)**

   The duration of the delay that will occur when this M-Code is executed. For example, if the spindle motor takes about three (3) seconds to reach full speed, this value should be at least three (3).

6. **Output Lines**

   Each output that is controlled by the M-Code is listed here with the desired state of the output.

7. **Add M-Code Button**

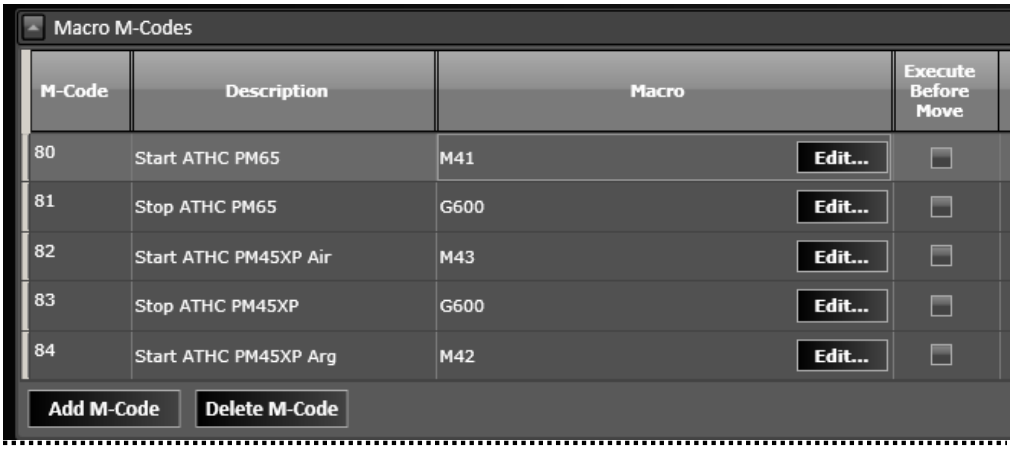
   Adds a new line to define an M-Code.

8. **Delete M-Code Button**

Deletes the highlighted M-Code definition.

13.20 Macro M-Codes

FlashCut can also use M-Codes to call a macro.



1. **M-Code**

   The number of the M-Code that you want to define.

2. **Description**

   A brief description of the action taken when this M-Code is executed.

3. **Macro**

   Click the Edit button to bring up the macro editor. Enter any number of G-Code lines. Whenever FlashCut encounters the M-Code in a G-Code program, these commands will be executed.

4. **Execute Before Move**

   Indicates the sequence of actions when there is a machine tool move command on the same program line as the M-Code. Check this box to execute the M-Code before the machine moves as commanded.

5. **Add M-Code Button**

   Deletes the highlighted M-Code definition.

6. **Delete M-Code Button**

   Deletes the highlighted M-Code definition.

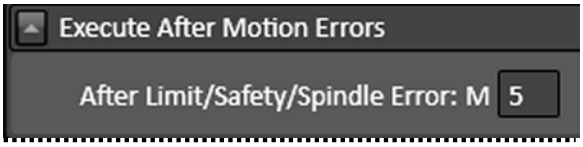
13.21 M-Code Execution

The M-Code execution option presents these M-Code execution configuration types:

- Execute After Motion Errors
- Execute at Start/End of Motion
- Execute During Ramping
- Execute While Connecting/Disconnecting

Any M-Code entered with these configuration types must be defined on the M-Code Definitions panel.

13.22 Execute After Motion Errors

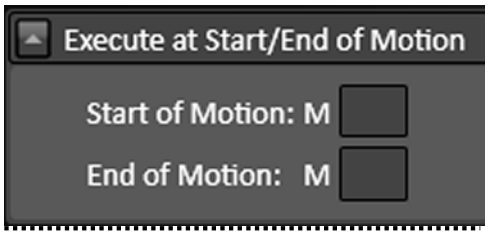


After Limit/Safety/Spindle Error – FlashCut executes the M-Code after an error occurs. This is useful in applications where it is safest to automatically turn off the spindle, vacuum and other devices whenever the signal generator detects an error.

Errors include limit switch tripped, safety switch tripped, or spindle stopped during thread cutting (lathe only).

When running a lathe, and especially when cutting threads, this mode is highly recommended. If an error occurs (e.g., a limit switch gets tripped), FlashCut immediately stops moving the tool. The spindle must then be stopped, or else there will be interference between the stationary tool and the spinning workpiece. The automatic M-Code should be configured to stop the spindle.

13.23 Execute at Start/End of Motion



A typical use for this feature is to turn on a warning light whenever machine tool is moving.

- Start of Motion – FlashCut executes this M-Code immediately before any motion begins (including starting a G-Code program, jogging, and so on).

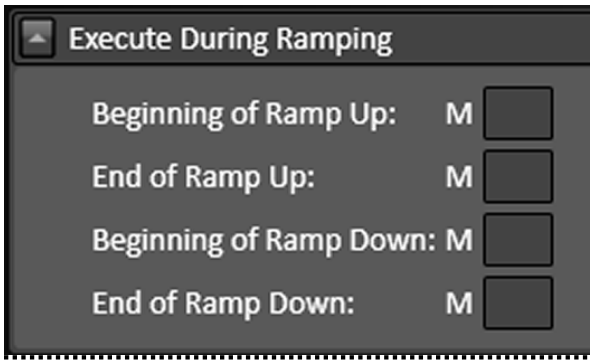
- End of Motion – FlashCut executes this M-Code immediately after motion stops.

NOTE

Motion includes actual machine moves as well as dwells.

13.24 Execute During Ramping

A typical use for this feature is to turn on/off an external device when the machine tool is accelerating or decelerating. FlashCut turns the device on/off only while running a G-Code file.



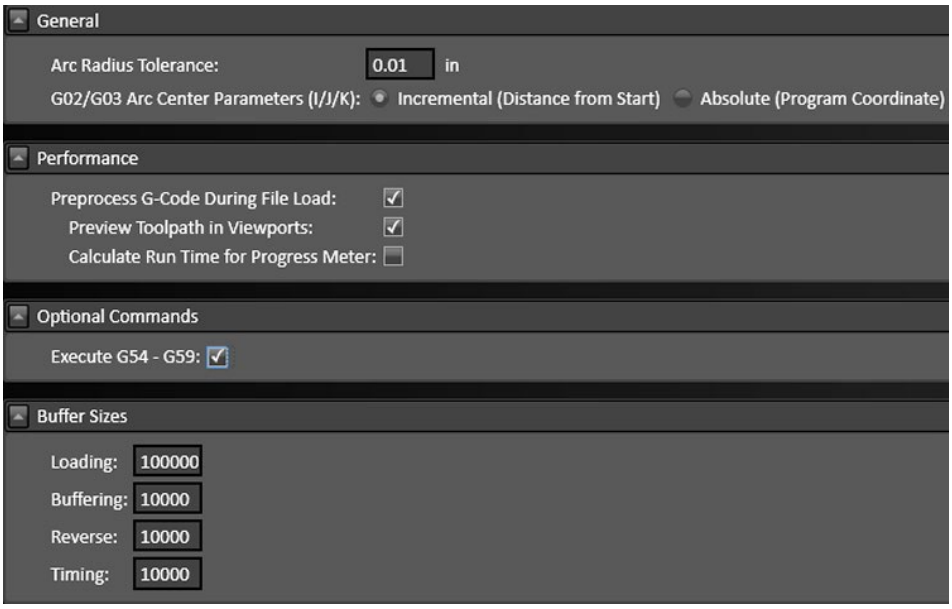
- Beginning of Ramp – FlashCut executes this M-Code when ramping up begins.
- End of Ramp Up – FlashCut executes this M-Code when ramping up ends.
- Beginning of Ramp Down – FlashCut executes this M-Code when ramping down begins.
- End of Ramp Down – FlashCut executes this M-Code when ramping down ends.

13.25 Execute While Connecting/Disconnecting

A typical use of this feature is to turn on and off a warning light indicating the machine tool is active.

- Connecting – FlashCut executes this M-Code whenever the software is connecting to the signal generator.
- Disconnecting – FlashCut executes this M-Code whenever the software is disconnecting from the signal generator.

13.26 G-Code



13.27 General

Arc Radius Tolerance – Sets the maximum allowable difference between the starting and ending radius for an arc move (G02 or G03). While loading files, FlashCut checks every arc to make sure the arc radius difference is within the tolerance specified here.

13.27.1 Performance

- Preprocess G-Code during File Load – If checked, FlashCut checks the G-Code file for errors whenever a file is opened or reset. If unchecked, FlashCut skips this step. When running very large files, you may want to uncheck this option to save time opening and resetting G-Code files.
- Preview Toolpath in Viewports – If checked, FlashCut displays a preview of the toolpath.
- Calculate Run Time for Progress Meter – If checked, FlashCut will estimate the run time for the program.

13.28 Optional Commands

When this option is selected, G-Code commands G54 through G59 will be executed.

13.29 Buffer Sizes

Sets G-Code buffer sizes: Loading, Buffering, Reverse, Timing.

13.30 I/O

- Input Lines
- Output Lines

13.31 Input Lines

The Input Line Setup configuration area allows you to configure each of the thirty-two (32) available input lines.

13.32 Set All Wiring

Sets the Wiring option for every line to either Normally Open or Normally Closed.

13.33 Line Setup

You may set the following options for each input line.

In addition, you may click one of the two (2) buttons to set each input line as normally open or normally closed.

- Line
- Description
- Function
- Axis
- Sub-axis
- Position
- Sensing Debounce
- Wiring

13.34 Line

Identifies the line number on the Signal Generator's Input connector.

13.35 Description

A brief explanation of the input line, used by FlashCut in various system messages.

13.36 Function

The function of the switch as follows:

- **Home/Limit** – The switch will be used for homing and limiting travel.
- **Home** – The switch will be used for homing only.
- **Limit** – the switch will be used for limiting travel only.
- **Safety** – The switch will stop motion immediately (e.g. a servo error line). The switch behaves the same as a Limit Switch, but does not have a Location associated with it.
- **Control** – The switch will be used for general control purposes (e.g. for tool length sensing or program zero sensing).
- **Feed Hold** – The switch will cause a feed hold to occur, just as if the Feed Hold button were clicked.
- **Feed Hold (Emergency)** – Same as Feed Hold, except that the operation uses the Emergency Ramp Rate specified in the Feedrates and Ramping table. See **Feedrates and Ramping**.
- **Start** – FlashCut will start running the current G-Code file, just as if the Start button were clicked.
- **Run File** – FlashCut will load the specified G-Code file and start running it. Not currently enabled.
- **Load File** – FlashCut will load the specified G-Code file. Not currently enabled.
- **Unused** – The input line is not used and will be ignored by the system.

NOTE

Run File and Load File are not currently enabled but are expected to be added to a future release.

1. **Axis**

   Identifies the location (by axis) for all limit and home switches.

2. **Sub-axis**

   If your machine uses two (2) motors to drive an axis (defined on the Machine>Axes panel), FlashCut will include two (2) sub-axes in the list of choices.

3. **Position**

   Identifies the direction, for a given axis, for all limit and home switches.

4. **Sensing Debounce**

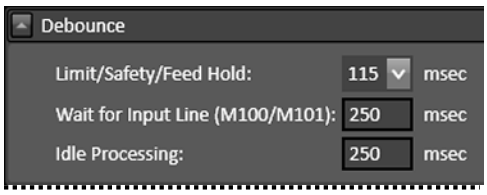
   For a switch to successfully reach a desired state, it must hold that state continuously for the debounce duration (in seconds).

5. **Wiring**

   Indicates whether the switch is wired normally open or normally closed. Use the Normally Open or Normally Closed button to set the wiring type for all input lines at once.

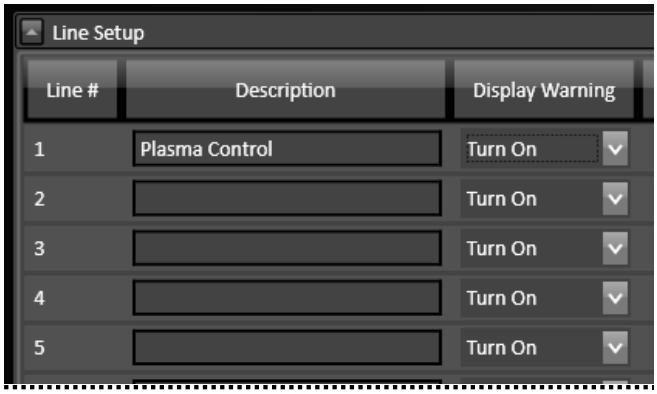
13.37 Debounce

This is the duration that a switch must be continuously tripped for FlashCut to recognize it as tripped.



- Limit/Safety/Feed Hold – Sets the duration in milliseconds for these switch functions.
- Wait for input Line – Sets the duration in milliseconds of the debounce for operations M100 and M101.
- Idle Processing – Sets the duration in milliseconds.

13.38 Output



- Line # - Line number on the Signal Generator's Output connector.
- Description – A brief explanation of the output line. FlashCut will display an on/off toggle button on the Auxiliary Control Panel for each output line with a Description entered.
- Display Warning – Indicates whether or not FlashCut will display a warning when the specified output line status changes. Options are Turn On, Turn Off, Always, and Never.

13.39 User Interface

13.39.1 Custom Buttons

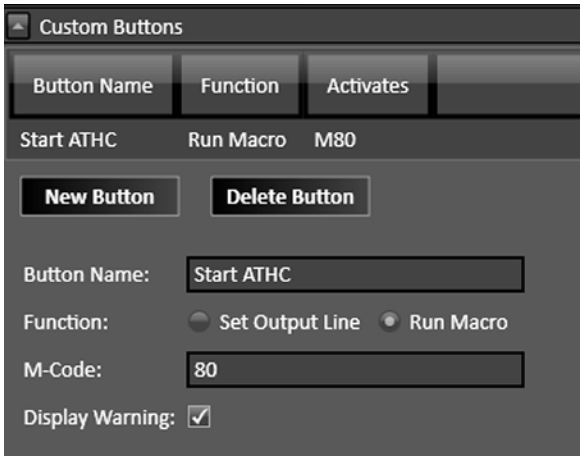
Custom buttons can be added to the CNC window for use during operations. These buttons can set an output line (see **Output Lines**) or run an existing macro (see **Macro M-Codes**).

The example to the right shows the properties of a custom button that launches an existing macro, M80 (start automatic torch height control).

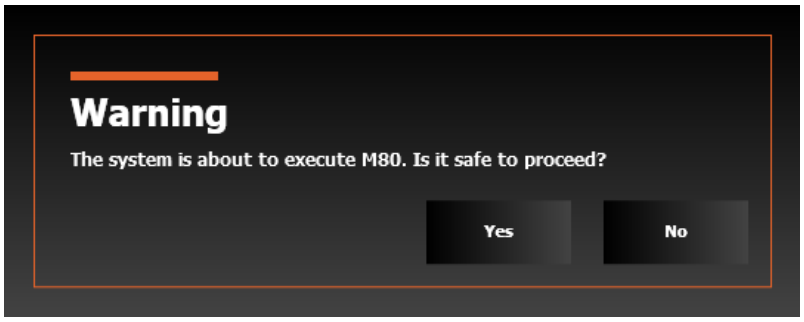
Note that the M-Code has already been defined and that the Display Warning checkbox has been selected.

Custom buttons appear in the CNC window. They are located in a panel to the left of the drawing workspace.

The button from example above is shown in the detail from the CNC window at right, circled in red.



Clicking the button triggers the warning dialog illustrated below.



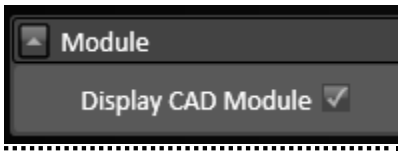
14.0 CAD

- General
- Features

14.1 General

14.1.1 Display CAD Module

When selected, the CAD module will be available in FlashCut.

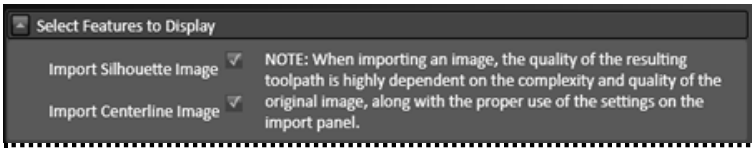


14.2 Features

14.2.1 Select Features to Display

The Select Features to Display configuration allows you to select the following items to display.

1. **Import Silhouette Image**
   This enables the use of the silhouette image import in the CAD module.



2. **Import Centerline Image**
   This enables the use of the centerline image import in the CAD module.

NOTE

When importing an image, the quality of the resulting toolpath is highly dependent on the complexity and quality of the original image, along with the proper use of the settings on the import panel.

14.3 CAM

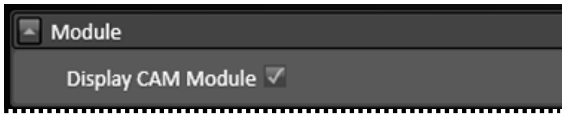
- General

- Features
- Post Processor

14.4 General

14.4.1 Module

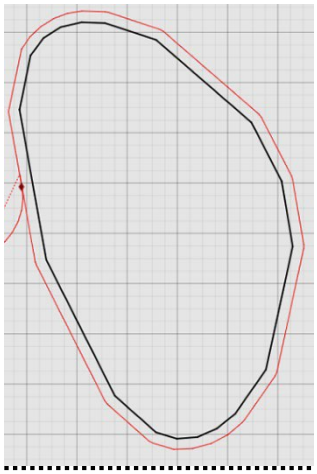
When selected the CAM module will be available in FlashCut.

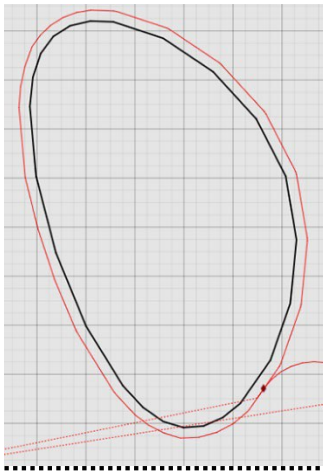


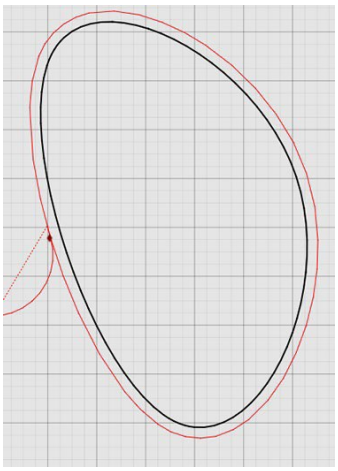
14.4.2 Toolpath Accuracy

Enter value here. Smaller values produce smoother curves. A value of 0.001 is recommended.

In the CAM toolpath examples below, the same closed curve drawn with the spline tool is shown at three (3) levels of toolpath accuracy 0.1, 0.01, and 0.001 (left to right).





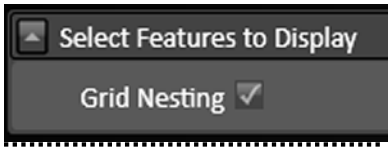


14.5 Features

Select Features to Display

Grid Nesting

This enables the use of grid nesting in the CAM module.

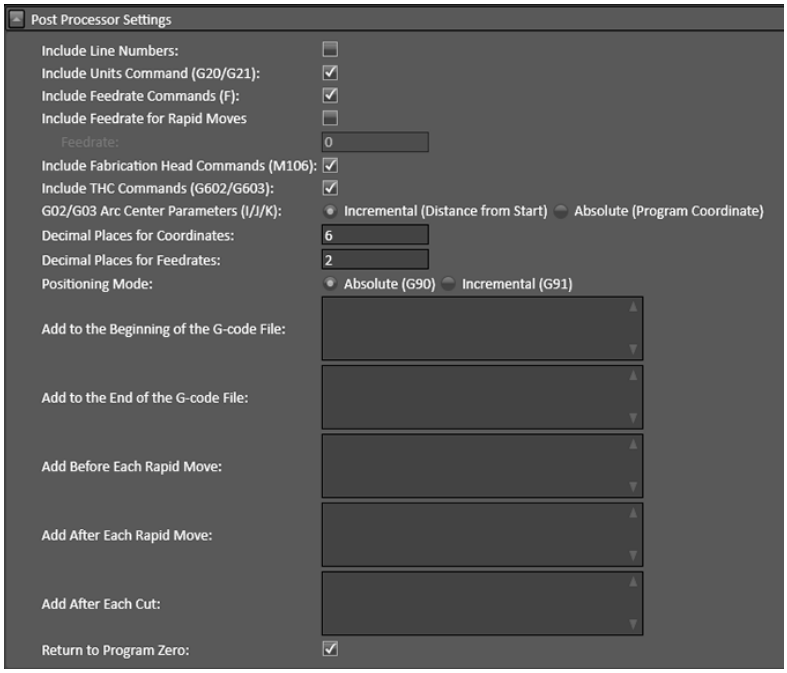


14.6 Post Processor

14.6.1 Post Processor Settings

1. **Include line numbers**

   Check this box if you want to number all of the G-Code lines in the program that's being created.



2. **Include units command**

   Check this box if you want to include the G20/G21 commands for Inch/Metric units respectively. The G-Code command will correspond with the units defined in the CAD/CAM settings menu.

3. **Include feedrate commands**

   Keep this box enabled: disabling this box is intended for internal use only. The feedrate command will no longer be included in the program if this box is disabled.

   Any G-Code command entered here will be added before all rapid moves (G00).

4. **Include feedrate for rapid moves**

   This will enable the use of the F feedrate command for a rapid move. Select the checkbox and enter the feedrate value to be used in the input field labeled feedrate.

5. **Include fabrication head commands**

   If selected, the M-106 command will be included in the G-Code tile. M-106 is the Fab Head change command: for example, changing from plasma Fab Head to the marker Fab Head.

6. **Include THC commands**

   For plasma fabrication heads only. Commands to enable and disable torch height control will be included in the G-Code file.

7. **G02/G03 Arch Center Parameters (I/J/K)**

Controls the definition of clockwise (G02) and counterclockwise (G03) circular arcs. Select either incremental (distance from start) or absolute (program coordinates).

8. **Decimal places for coordinates**

   Enter the number of decimal places to be used for feedrates.

9. **Decimal placed for feedrates**

   Enter the number of decimal places to be used for feedrates.

10. **Positioning mode**

    Select either absolute or incremental.

11. **Add to the beginning of the G-Code file**

    Any G-Code command entered here will be added at the beginning of all G-Code programs generated from the CAM module.

12. **Add to the end of the G-Code file**

    Any G-Code command entered here will be added at the end of all G-Code programs generated from the CAM module.

13. **Add before each rapid move**

    Any G-Code command entered here will be added before all rapid moves (G00).

14. **Add after each rapid move**

    Any G-Code command entered here will be added after rapid moves (G00).

15. **Add after each cut**

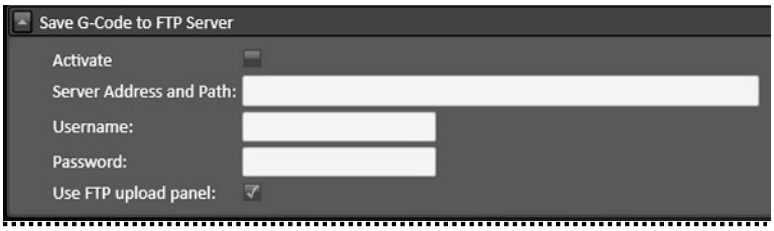
    Any G-Code command entered here will be added after each cut.

16. **Return to program zero**

    When selected, this feature is enabled.

14.6.2 Save G-Code to FTP Server

Use Save G-Code to FTP Server to instruct FlashCut to upload your G-Code file to an FTP server each time the file is saved.



14.6.3 Activate

Enable the Activate checkbox to upload your G-Code file to an FTP server each time each time the user clicks the cut button.

14.6.4 Server Address and Path

Enter the URL for the server including the sub-folder paths.

14.6.5 Username

Enter the username for the FTP account in the Username field.

14.6.6 Password

Enter the password for the FTP account in the Password field. FTP account password are typically case sensitive.

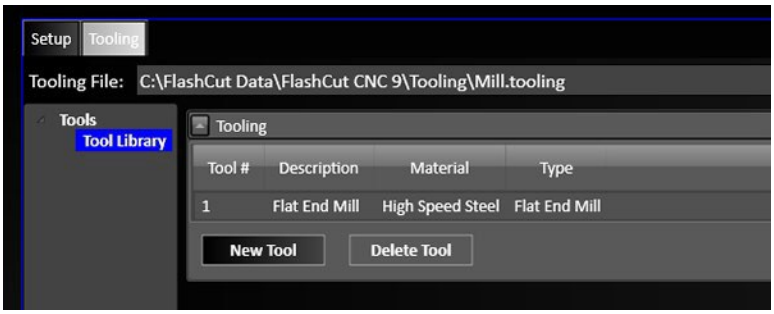
14.6.7 Use FTP upload panel

If Use FTP upload panel is checked: an FTP upload panel appears in the parameters pane with various control options.

If Use FTP upload panel is unchecked: the file is uploaded with a default filename (e.g. gcode.nc).

14.7 Tooling Configuration Tab

When the tab is selected, the tooling file setup window and Tool Library are visible.

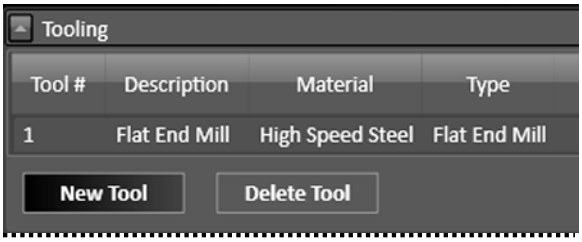


14.8 Tool Library

Tooling

The Tool Library stores information about tools used by specific Fab Heads, e.g., mills. Select a tool to edit its parameters. The following parameters are displayed in the list view.

- **Tool #** - Used throughout the software to identify the tool (e.g. in the M06 tool change command).
- **Description** – Used throughout the software to describe the tool.
- **Material** – Tool materials include high speed steel, carbon steel, uncoated carbide, coated carbide, polycrystalline CBN, and polycrystalline diamond.

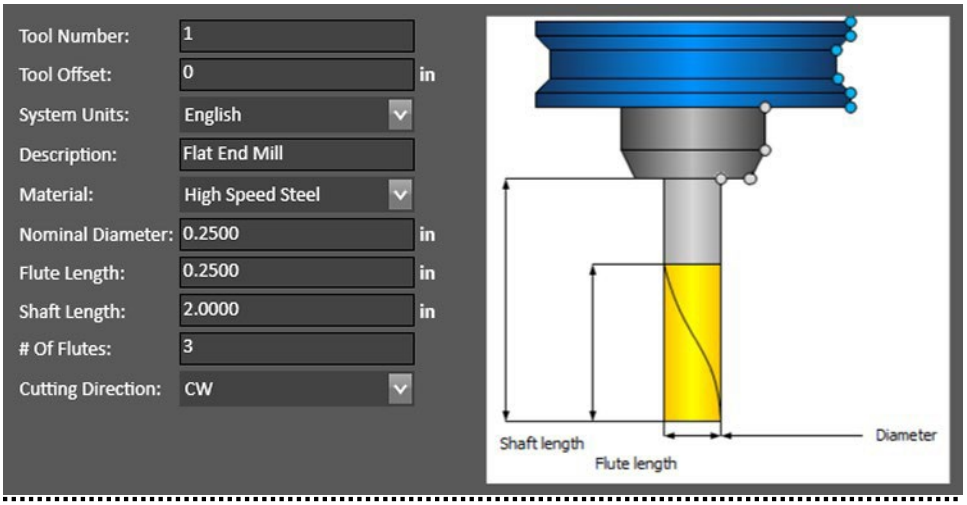


- **Type** – The following types of tools can be defined; Ball End Mill, Flat End Mill, Round Edge End Mill, Tapered Edge End Mill, Creasing Wheel, Dual sided Knife, Rotary Knife, and Straight Knife.

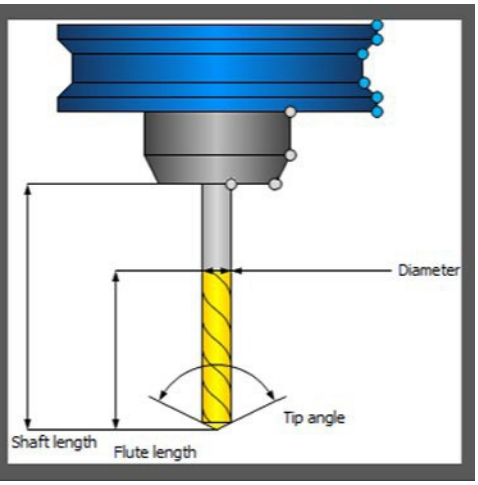
To add a tool to the library, select New Tool. To remove a tool from the library, select Delete Tool. When adding a tool, you must define its parameters before it can be used.

Each tool must be given a unique Tool Number. The Tool Offset and the specific dimensions for each tool, can be defined using either English or metric System Units. This definition is independent of the General **System Units** setting.

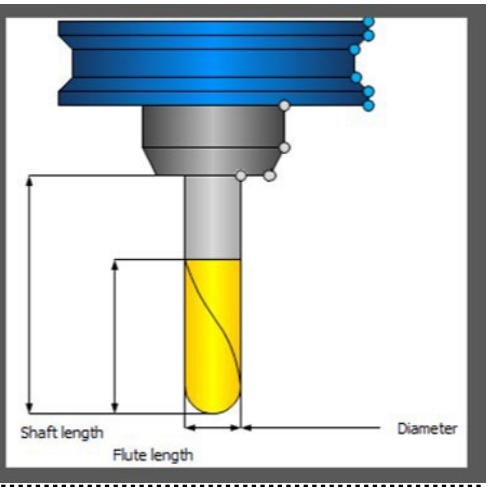
Sample tool configurations are shown on the following pages.



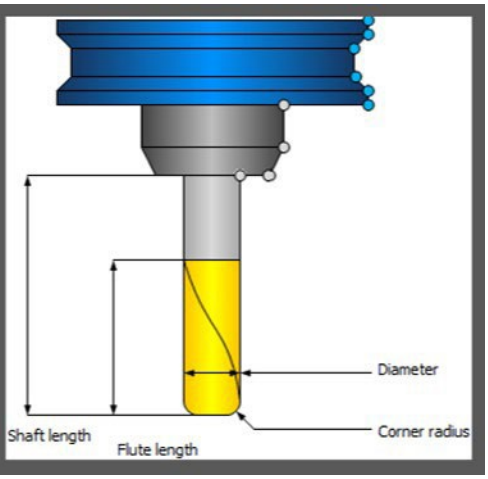
Tool Number:	3	
Tool Offset:	0	in
System Units:	English	
Description:	Drill	
Material:	Carbon Steel	
Nominal Diameter:	0.3125	in
Flute Length:	1.9000	in
Shaft Length:	3.7500	in
# Of Flutes:	2	
Cutting Direction:	CW	
Tip Angle:	118	°



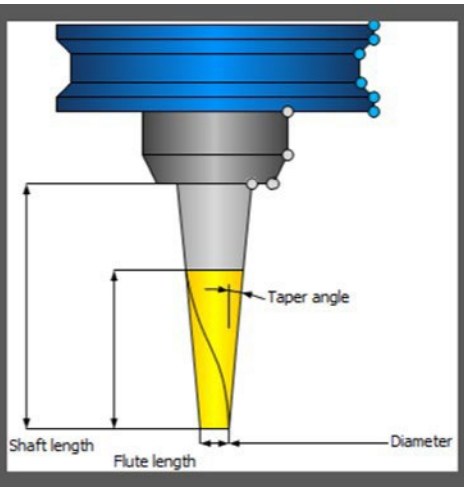
Tool Number:	2	
Tool Offset:	0	in
System Units:	English	
Description:	Ball End Mill	
Material:	Uncoated Carbide	
Nominal Diameter:	0.5000	in
Flute Length:	2.0000	in
Shaft Length:	4.0000	in
# Of Flutes:	2	
Cutting Direction:	CW	



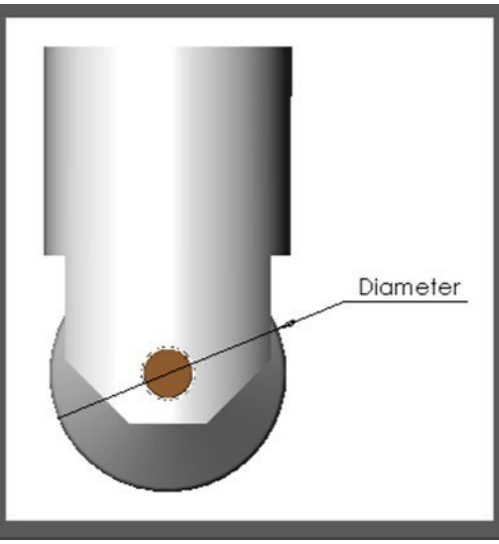
Tool Number:	4	
Tool Offset:	0	in
System Units:	English	
Description:	Round Edge End Mill	
Material:	Uncoated Carbide	
Nominal Diameter:	0.3125	in
Flute Length:	0.8125	in
Shaft Length:	2.5000	in
# Of Flutes:	4	
Cutting Direction:	CW	
Corner Radius:	0.045	in



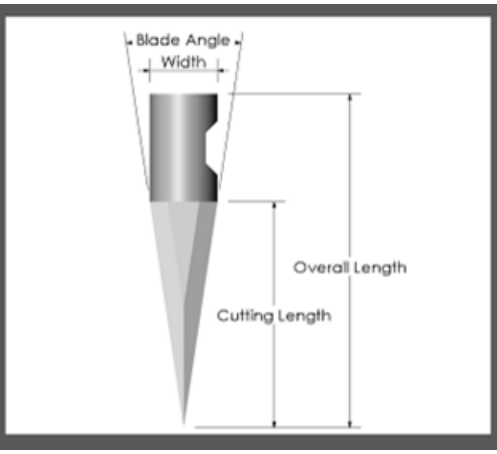
Tool Number:	5	
Tool Offset:	0	in
System Units:	English	
Description:	Tapered Edge End Mill	
Material:	High Speed Steel	
Nominal Diameter:	0.3750	in
Flute Length:	0.5000	in
Shaft Length:	2.5000	in
# Of Flutes:	3	
Cutting Direction:	CW	
Angle:	20	°

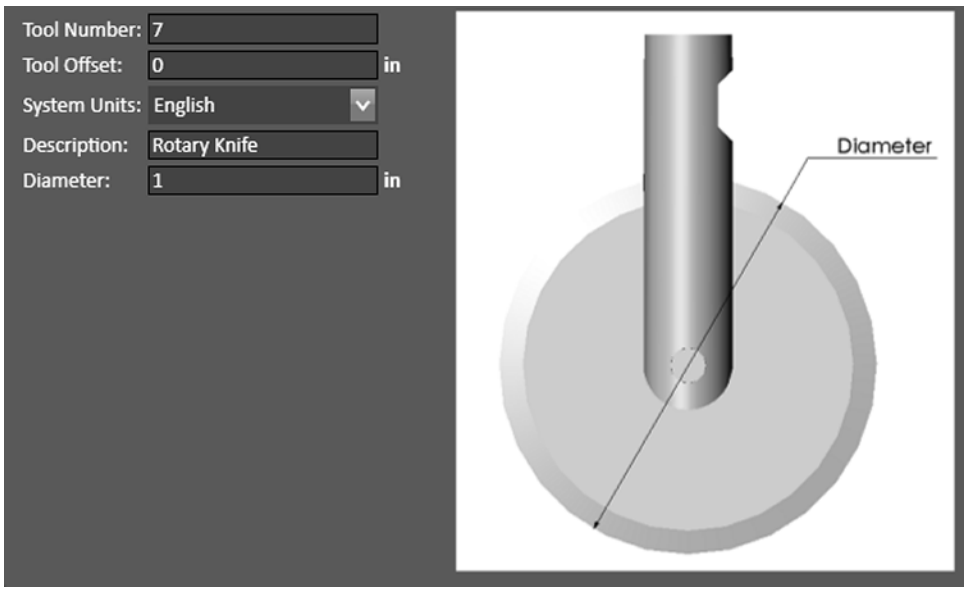


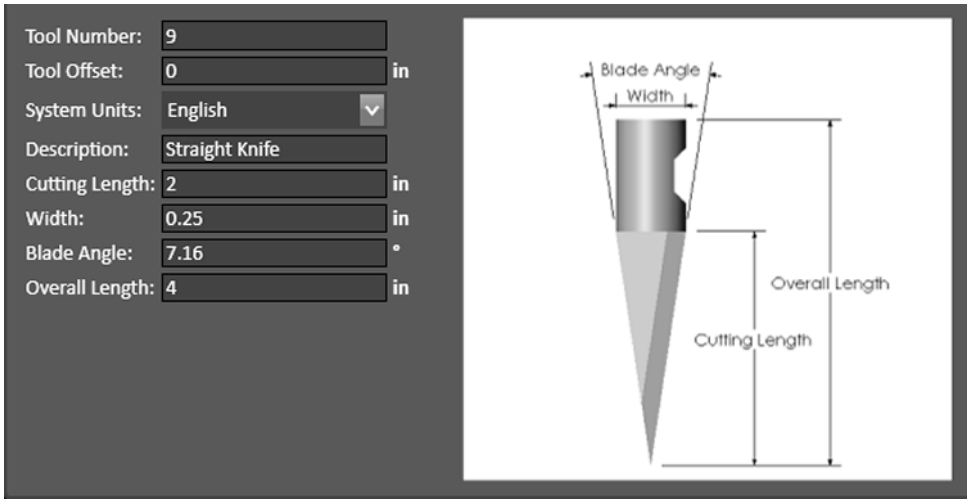
Tool Number:	6	
Tool Offset:	0	in
System Units:	English	
Description:	Creasing Wheel	
Diameter:	2	in



Tool Number:	8	
Tool Offset:	0	in
System Units:	English	
Description:	Dual Sided Knife	
Cutting Length:	2	in
Width:	0.25	in
Blade Angle:	7.16	°
Overall Length:	4	in







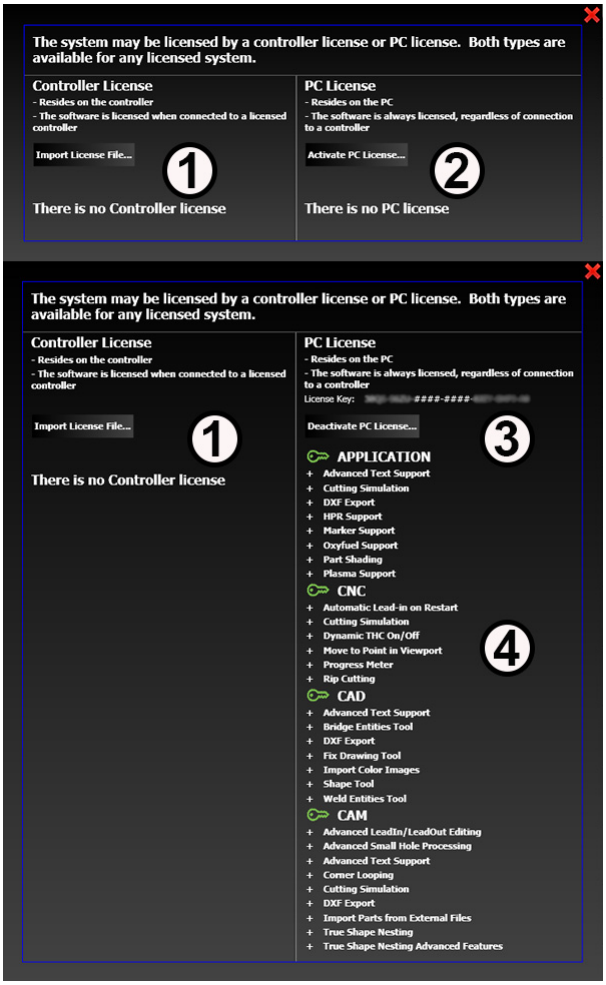
14.9 Licensing FlashCut

Click the License tab to display the FlashCut license management window.

There are two (2) forms of licensing in FlashCut: a license file for the controller and a license key for the computer connected to the controller. Both licenses can be managed through FlashCut enabled by the currently installed license and can also be reviewed in the window.

NOTE

An internet connection may be required when upgrading between versions to activate an existing PC license.



1. Import License File
2. Activate PC License
3. Deactivate PC License
4. Licensed Features List

14.10 License Management Window

14.10.1 Import License File

If you've received a license file, you may import the file into this software to license it. Follow these steps to import your license into your software.

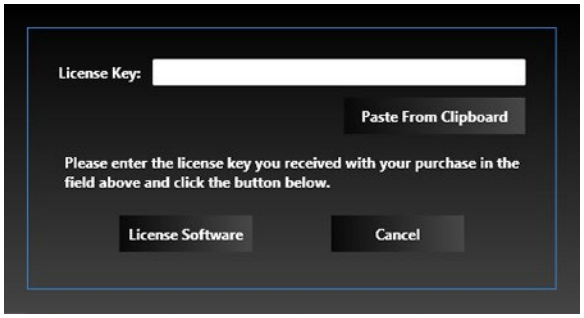
1. When you receive a license file, copy it to an appropriate location on your computer.

2. Click the Import License File…button.
3. FlashCut displays and Open dialog box.
4. Browse to the location of the file, select it, and click OK.
5. FlashCut installs the license and displays this notice to confirm that the license has been activated.
6. Click OK. The license will be loaded into your controller the next time you connect to it.

14.11 Activate PC License

If you've received a license key in an email message, you may license your software by typing or pasting the license key into a dialog box. Follow these steps to license your software with a license key from an email message.

1. Click the Activate PC License…button.
2. FlashCut displays a License Your Software dialog box.

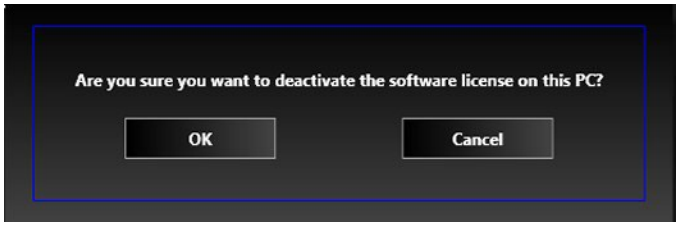


3. If you've copied the license key form the email you received, click the Paste From Clipboard button. FlashCut pastes the license key in the License Key field. Otherwise, you may type the license key in the field yourself.
4. Click the License Software button.
5. FlashCut installs license and displays a notice to confirm that the license has been activated. The license will be loaded into your controller the next time you connect to it.
6. The license will be loaded into your controller the next time you connect to it.

14.12 Deactivate PC License

You may deactivate the license on our computer so it can be used on a different computer. Follow these steps to deactivate the license on your computer.

1. Click the Deactivate PC License…button.
2. FlashCut displays a confirmation dialog box.



3. Click OK to deactivate the license.

14.13 Licensed Features List

This list shows which advanced or a la carte features of FlashCut are enabled by the installed license key. For an overview of FlashCut features, see **Program packages and features.**

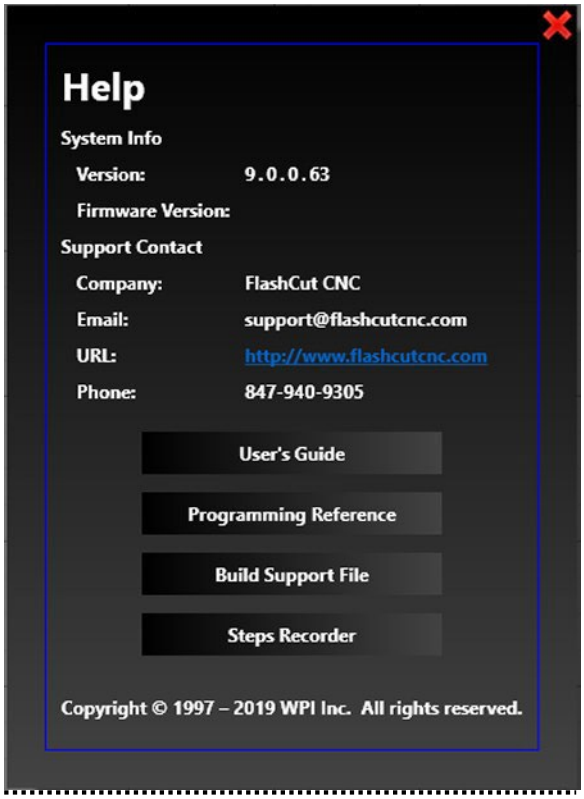
14.14 Getting Help

14.14.1 Help

Selecting the Help icon for any window brings up the following Help screen.

This screen provides information that can help you learn or troubleshoot the software and that can help FlashCut technical support assist you on a support call.

- Version
- Firmware Version
- Support Contact
- User's Guide
- Programming Reference
- Build Support File
- Build Support File



14.14.2 Version

The software revision identification number that is currently installed on the PC.

14.14.3 Firmware

The firmware revision identification number that is currently loaded on the signal generator.

14.14.4 Support Contact

Displays the support contact information for your version of the software.

14.14.5 User's Guide

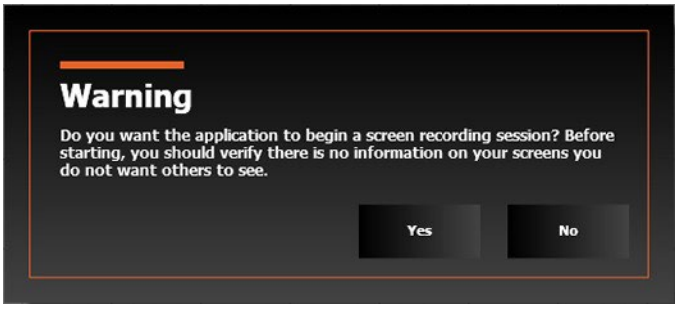
Displays this User's Guide in PDF format.

14.14.6 Programming Reference

Displays the FlashCut Programming Reference in PDF format. Requires a PDF viewer such as Adobe® Acrobat.®.

14.14.7 Steps Recorder

Begins a screen recording session. The following waning dialog is displayed.



While a screen recording session is in progress, a red square icon will appear in the upper right fo the program window. Select the icon to Stop Recording.

15.0 Maintenance and Troubleshooting

15.1 Lubrication

Lubricate the bearing surfaces and the ball screws every twelve (12) hours. Use a thin Lithium spray or a 30 WT oil lubricant.

15.2 Periodic Maintenance

In addition to the following periodic routines, regularly clean the machine and surrounding area to maintain the environment.

15.2.1 Daily

1. Clean the machine and lubricate unpainted surfaces with a 30 WT oil lubricant. Wipe off any excess and buff with a dry polishing cloth. This will reduce the likelihood of rust forming.
2. Inspect the overall machine for damage and loose or worn parts.
3. Empty the slag pan(s).

15.2.2 Weekly

1. Inspect the overall machine for damage and loose or worn parts.
2. Verify all the electrical connectors are fitted correctly and are not loose.
3. Verify all the motor couplers are connected and the screws are tight.
4. Wipe the linear guides for the X- and Y-Axis with a dry rag. Apply light coating of oil.

15.2.3 Quarterly

1. Lubricate the X- and Y-Axis racks.
2. Use Molly lubricant.

15.2.4 Annually

1. Lubricate the X- and Y-Axis bearings.

2. Use premium grade lithium base, extreme-pressure grease such as Shell Alvania EP-1.

15.3 Position of Home Switches

The home switches are activated by proximity to steel items and can be tested by placing a screwdriver or similar tool on the activation face. When activated, an LED should light. If the LED does not come on, the switch or wiring is faulty.

15.4 Oiling the Machines

The machine is provided with a central oiler. Do not over-lubricate the machine, as the excess oil attracts dirt. It is recommended that one pump of the oiler once a month will be sufficient to keep the machine lubricated. When the oil tank needs filing, top up with a good quality SAE 30 WT oil.

15.5 Troubleshooting

Issue	Solution
Machine will not start	1. Check the start switch is being pressed fully in. 2. Check that the red emergency switch is fully released. 3. Check the electrical power cord is plugged into the power outlet. 4. Check the electrical supply is on (reset the breaker). 5. With the power disconnected from the machine, check that the wiring to the plug is correct. Verify that the rubber insulation is stripped enough and does not cause a bad connection. Verify that all the screws are tight.
Stepper motor tries to start but will not turn	1. With the power disconnected from the machine, try to turn the relevant motor by hand. If the motor will not turn, check the reason for jamming. 2. Motor faulty. Replace the motor. 3. Check that the voltage supplied to the machine is 220V.

Issue	Solution
Squeaking Noise	1. Check the bearings. 2. Check pulleys and belt for correct tension.
Machine Vibrates	1. The machine is not level. 2. Re-level the machine.
Machine will not home	1. Verify the home position sensors are connected. 2. Verify the home position sensors are not damaged or out of adjustment. 3. Verify the controller parameters are correct. 4. Press OK after completing a project and verify the cutting head returns to the home position.
Jobs are not cut consistently	1. Verify the motor drive belt is tight and not damaged. Replace the belt if is damaged. 2. Verify the drive couplings are tight and undamaged. 3. Verify the slider bearings are tight undamaged. Tighten if loose; replace if damaged.
Inaccurate position of the cutting head	1. Verify the drive screw and the bearing rails are clean and lubricated. 2. Verify the gantry head movement not too fast. 3. Verify all the bearings and motor fixing bolts are tight. 4. Verify the input voltage is correct. It must be 220V. 5. Verify there are no obstructions on the gantry.

Issue	Solution
The Controller Screen is blank, dull, or flickers.	1. Check for blown fuses. 2. If the cable is damaged, replace the damaged parts. 3. If the power supply is damaged, replace the power supply. 4. If the controller is damaged, replace the controller. **NOTE** ***The power must only be 24V.***
The gantry or cutting torch will not function	1. The controller cable is loose. 2. The drive wires are loose or damaged. 3. The controller is damaged. 4. The Emergency Stop (E-Stop) button is depressed (in the down position). Reset the E-Stop button.

16.0 Warranties

Dealer Machinery Warranty

New woodworking machines sold by Laguna Tools carry a two-year warranty effective from the date of dealer invoice to customer/end-user. Machines sold through dealers must be registered with Laguna Tools within thirty (30) days of purchase to be covered by this warranty. Laguna Tools guarantees all new machines sold to be free of manufacturers' defective workmanship, parts, and materials. We will repair or replace, without charge, any parts determined by Laguna Tools, Inc. to be a manufacturer's defect. We require that the defective item/part be returned to Laguna Tools with the complaint. The end-user must request a Return Material Authorization (RMA) number from Customer Service. Include the RMA number with any and all returned parts/components requesting warranty coverage*. Any machines returned to Laguna Tools must be returned with packaging in the same manner in which it was received. A part or blade is being returned must have adequate packaging to ensure it is not damaged during shipping. In the event the item/part is determined to be damaged due to lack of maintenance, cleaning, or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges. This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, lack of or inadequate dust collection, misuse/abuse or damage caused where repair or alterations have been made or attempted by others.

* The issue of an RMA number is for reference only; it DOES NOT indicate acceptance of the warranty claim.

CNC Limited Warranty

New CNC machines sold by Laguna Tools carry a one-year warranty effective from the date of shipping. Laguna Tools guarantees all new machines sold to be free of manufacturers' defective workmanship, parts, and materials. We will repair or replace, without charge, any parts determined by Laguna Tools, Inc. to be a manufacturer's defect. If the defective item/part is determined to be damaged due to lack of maintenance, cleaning or misuse/abuse, the customer will be responsible for the cost to replace the item/part, plus all related shipping charges. This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, lack of or inadequate dust collection, misuse/abuse or damage caused where repair or alterations have been made or attempted by others.

Laguna Tools, Inc. is not responsible for additional tools or modifications sold or performed (other than from/by Laguna Tools, Inc.) on any Laguna Tools, Inc. woodworking machine. Warranty may be voided upon the addition of such described tools and/or modifications, determined on a case-by-case basis.

Software purchased through Laguna Tools, Inc., is not covered under this warranty and all technical support must be managed through the software provider. Normal user alignment, adjustment, tuning, and machine settings are not covered by this warranty. It is the responsibility of the user to understand basic woodworking machinery settings and procedures and to properly maintain the equipment in accordance with the standards provided by the manufacturer.

Parts under warranty are shipped at Laguna Tools, Inc.'s cost either by common carrier, FEDEX ground service, or a similar method. Technical support to install replacement parts is primarily provided by phone, fax, e-mail or Laguna Tools Customer Support Website. The labor required to install replacement parts is the responsibility of the user. Laguna Tools is not responsible for damage or loss caused by a freight company or other circumstances not in our control. All claims for loss or damaged goods must be notified to Laguna Tools within twenty-four (24) hours of delivery.

Please contact our Customer Service Department for more information. Only NEW machines sold to the original owner are covered by this warranty.

For warranty repair information, call 1-800-332-4094.

No Modifications Allowed or Sold

Laguna Tools, Inc. is not responsible for additional tools or modifications sold or performed (other than from/by Laguna Tools, Inc.) on any Laguna Tools, Inc. woodworking machine. Warranty may be voided upon the addition of such described tools and/or modifications, determined on a case-by-case basis. Normal user alignment, adjustment, tuning, and machine settings are not covered by this warranty. It is the responsibility of the user to understand basic woodworking machinery settings and procedures and to properly maintain the equipment in accordance with the standards provided by the manufacturer. Parts, under warranty, are shipped at Laguna Tools, Inc.'s cost either by common carrier, FEDEX ground service or a similar method. Technical support to install replacement parts is primarily provided by phone, fax, e-mail, or Laguna Tools Customer Support Website. The labor required to install replacement parts is the responsibility of the user. Laguna Tools is not responsible for damage or loss caused by a freight company or other circumstances not in our control. All claims for loss or damaged goods must be notified to Laguna Tools within twenty-four (24) hours of delivery. Please contact our Customer Service Department for more information. Only new machines sold to the original owner are covered by this warranty.

For warranty repair information call 1-800-332-4094.

Laguna Tools Warranty

WARRANTY & REGISTRATION

THANK YOU!

Welcome to the Laguna Tools® group of discriminating woodworkers. We understand that you have a choice of where to purchase your machines and appreciate the confidence you have in the Laguna Tools® brand.

Through hands-on experience, Laguna Tools® is constantly working hard to make innovative, precision products. Products that inspire you to create works of art, are a joy to operate, and encourage your best work.

Laguna Tools®
Imagination, Innovation, and Invention at Work

WARRANTY & REGISTRATION

Every product sold is warranted to be free of manufacturers' defective workmanship, parts, and materials. For any questions about this product, the intended use or what it was designed for, customer service, or replacement parts, please contact our customer service department:

Laguna Tools® Customer Service
744 Refuge Way, Grand Prairie, Texas 75050, USA
1-800-234-1976
customerservice@lagunatools.com
www.lagunatools.com/why/customer-service/
8AM. to 5PM PST, Monday through Friday

For warranty claims or to report damage upon receiving – please reach out to our warranty department:

Laguna Tools® Warranty Service
744 Refuge Way, Grand Prairie, Texas 75050, USA
1-800-332-4049
customerservice@lagunatools.com
www.lagunatools.com/rpolicies/warranty
8AM to 5PM PST, Monday through Friday

REGISTRATION

To prevent voiding this warranty, all products sold must be registered within thirty (30) days of receiving the product. Registering the product will enable the original purchaser to receive notifications about important product changes, receive customer service, and be able to file a warranty claim against defective workmanship, parts, or materials.

REGISTER HERE!

https://lagunatools.com/policies/warranty/

WHO IS COVERED

The applicable warranty covers only the initial purchaser of the product from the date of receiving the product. To file such claims, the original purchaser must present the original receipt as proof of purchase.

WHAT IS COVERED

The warranty covers any defects in the workmanship of all parts and materials that make up the machine unless otherwise specified. Any part, determined by Laguna Tools®, to have a defect will be repaired or replaced (and shipped), without charge. The defective item/part must be returned to Laguna Tools® with the complaint and proof of purchase in the original packaging that it was received in. In the event the item/part is determined to be not covered by this warranty, the customer will be responsible for the cost to replace the item/part and all related shipping charges.

WARRANTY LIMITATIONS

This limited warranty does not apply to natural disasters, acts of terrorism, normal wear and tear, product failure due to lack of maintenance or cleaning, damage caused by accident, neglect, or lack-of inadequate dust collection. The warranty may be voided against proof of misuse/abuse, damage caused where repair or alterations have been made or attempted by others, using the product for purposes other than those described as intended use (unless with consent by Laguna Tools®), modification to the product, or use with an accessory that was not designed for the product. It is the responsibility of the user to understand basic woodworking machinery settings and procedures and to properly maintain the equipment in accordance with the standards provided in this manual.

LENGTH OF WARRANTY

All new machines and optional accessories sold through an authorized dealer carry a two-year warranty effective the date of receiving the product. Machines sold for either commercial or industrial use have a one-year warranty. Wearable parts like throat plates, bandsaw guides, etc., have a ninety-day warranty.

Table A-1 Warranty Lengths

2 Year – New Machines Sold Through an Authorized Dealer
2 Year – Accessories Sold as Machine Options (excluding blades)
1 Year – Machines Sold for Commercial or Industrial Use
1 Year – Blades and Accessories outside of Machine Options
90 Days – Wearable Parts

Aside from being free of defects upon receiving, consumable parts, like cutters and abrasives, are not covered by this warranty unless otherwise stated by Laguna Tools®. These parts are designed to be used at the expense of the operator and are available for replacement or inventory purchase. The determination of a consumable part will be made on a case-by-case basis by Laguna Tools®.

SHIPPING DAMAGE

Laguna Tools® is not responsible for damage or loss caused by a freight company or other circumstances not in the direct control of Laguna Tools®. All shipping-related claims for loss or damage goods must be made to Laguna Tools within twenty-four hours of delivery.

HOW TO RECEIVE SUPPORT

To file a warranty-claim please contact the warranty department at 1-800-234-1976. To receive customer service or technical support please contact the customer service department at 1-800-332-4049. Parts, under warranty, are shipped at the expense of Laguna Tools® either by common carrier, FedEx ground services or similar method. Technical support to install replacement parts is primarily provided by phone, fax, email, or the Laguna Tools Customer Support Website.

LAGUNA

Laguna Tools, Inc.
744 Refuge Way
Grand Prairie, TX 75050
800-234-1976
www.lagunatools.com