FLASHCUT CNC

CONTROL MADE SIMPLE

®

FlashCut CAD/CAM and CNC

Version 10

User's Guide

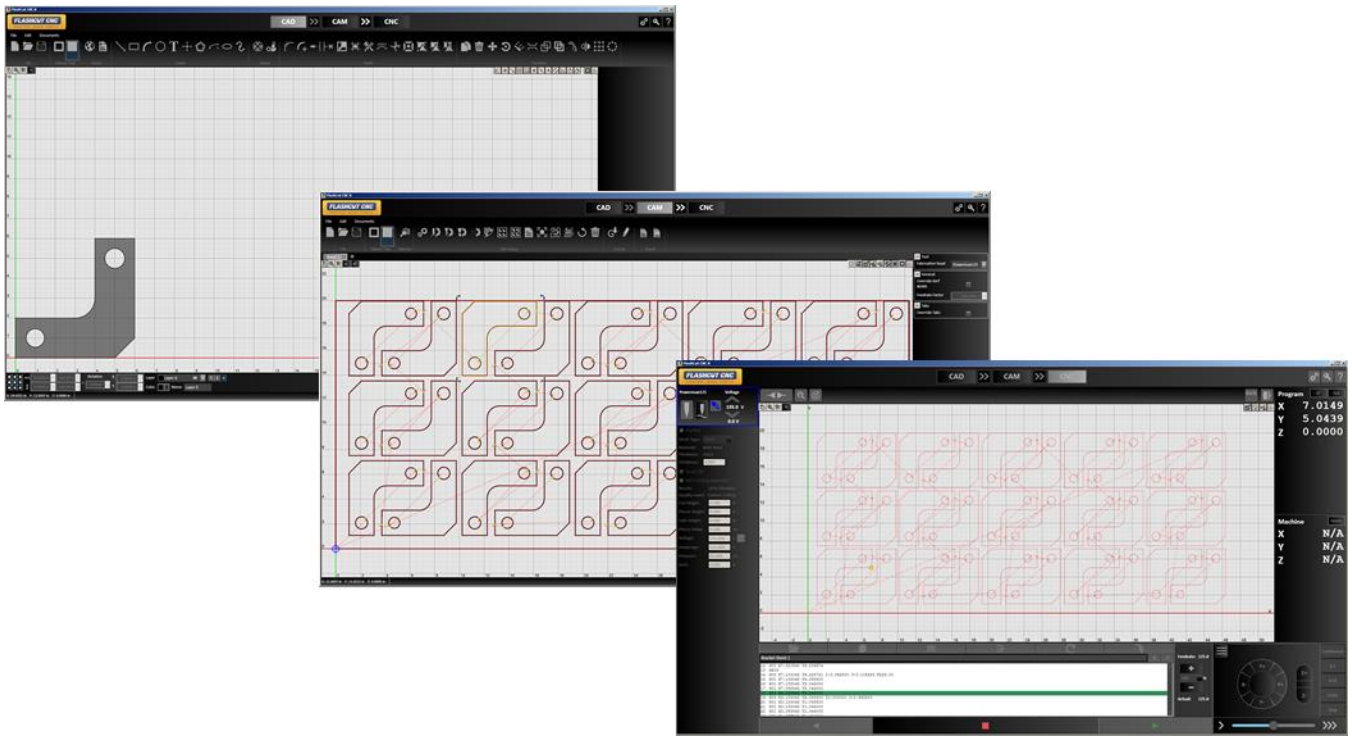


Table of Contents

NOTICES ..... **X**

About this document x
Disclaimer x
Safety and usage guidelines x
Technical support xi
Website ..... *xi*
Dealer support ..... *xi*
Email ..... *xi*
Phone/fax support ..... *xii*

INTRODUCING FLASHCUT® CNC **1**

System Requirements 1
Installing the FlashCut CNC Software 1
Preparing to Install the Software *1*
Installing the Software *1*
Updating the Setup File *3*
Installing the USB Driver 3

GETTING STARTED **4**

Navigating Through CAD CAM CNC 4
Apply / Cancel 4
Running in Evaluation Mode 5
Licensing FlashCut 5
Controller License *5*
Import License File 5
PC License *6*
Activate PC License 6
Deactivate PC License 7
Licensed Features List *7*
Demonstration License *7*
Educational License *8*
Temporary License *8*

CONFIGURATION **9**

Configuration Files 10
Setup File Selection 10
Tooling File Selection 10
Setup Configuration Menu 11
System 11
General *11*
General 11
System Units 11
Controller *11*
Controller 11
Motor Signals 12
Signal State Preferences 14
Safety *15*
Automatic Safety Response 15
Safety Interlocks 15
Security *16*
Security 16
Preferences 17
Language *17*
Preferred Language 17

Jogging ... *18*
Jog and Point Move Rates ... 18
Jog Distances ... 18
General ... 18
Point Move ... *18*
General ... 19
Display ... *19*
Axis Display ... 19
Current Position ... 19
Pendant ... *19*
General ... 19
USB Interface Board ... *21*
General ... 21
Messages ... *22*
MACHINE ... 23
Fabrication Heads ... *23*
Fabrication Heads ... 23
Plasma ... 23
Oxyfuel ... 32
Waterjet ... 36
Laser ... 41
Laser Pointer ... 41
Marker ... 43
Mill ... 46
Tangential Knife ... 51
Hotwire ... 55
Cut Charts ... 57
Fabrication Head Change ... *57*
Pause on Fabrication Head Change ... 57
Axes ... *58*
Motor Line/Axis Mapping ... 58
Axis Settings ... 59
Feedrates and Ramping ... 61
Advanced Axis Definition ... 64
Drive Parameters ... *64*
Axis Drive Parameters ... 64
Rotary Axes ... *70*
Rotary Axis Arrangement ... 70
General ... 71
Reference Points ... *72*
Reference Points ... 72
Fixture Offsets ... *72*
Fixture Offsets ... 72
Homing ... *73*
Axis Settings ... 73
Homing Initiated by User ... 75
Homing Initiated While Running G-Code ... 75
Automatic Macros ... 75
General ... 76
Dynamic Ventilation ... *77*
Tool Changer ... *79*
Tool Changer ... 79
Tool Rack Positions ... 79
SENSING ... 80
Tool Length Sensing ... *80*
Tool Length Sensor ... 80
Workpiece Sensing ... *81*
Round Pipe ... 81
PROGRAMMING ... 82

M-Code Definitions ... *82*
Output Line M-Codes ... 82
Macro M-Codes ... 83
M-Code Execution ... *83*
Execute at Start/End of Motion ... 84
Execute While Connecting/Disconnecting ... 84
G-Code ... *84*
General ... 85
Performance ... 85
Optional Commands ... 85
Buffer Sizes ... 85
I/O ... 85
Input Lines ... *85*
Set All Wiring ... 85
Line Setup ... 85
Debounce ... 87
Output Lines ... *88*
USER INTERFACE ... 89
Custom Buttons ... *89*
CAD ... 90
General ... *90*
Features ... *90*
Select Features to Display ... 90
CAM ... 91
General ... *91*
Module ... 91
Geometry Processing ... 91
Pipe Beveling ... 91
Features ... *92*
Select Features to Display ... 92
Grid Nesting ... 92
Save G-Code to FTP Server ... 92
Post Processor ... *93*
Basic Post Processor Settings ... 93
Advanced Post Processor Settings ... 94
TOOLING CONFIGURATION TAB ... 96
Tool Library ... *96*
Tooling ... 96

CAD/CAM COMMON INTERFACE ELEMENTS ... 100

MENU BAR ... 100
File Menu ... *100*
New (Ctrl+N) ... 100
Open (Ctrl+O) ... 101
Close ... 101
Close All ... 101
Save (Ctrl+S) ... 101
Save As… ... 101
Save All ... 101
Export ... 101
Edit Menu ... *103*
Undo (Ctrl+Z) ... 103
Redo (Ctrl+Y) ... 103
Settings… ... 103
Cut (Ctrl+X) ... 104
Copy (Ctrl+C) ... 105
Paste (Ctrl+V) ... 105
Select All (Ctrl+A) ... 105
Document Tabs ... *105*

PART SHADING AND GRID TOOLS 105
PARAMETERS AREA 105
STATUS BAR 105

FLASHCUT CAD 107

CAD RIBBON 107
File 108
New Document (Ctrl+N) 108
Open… (Ctrl+O) 108
Save (Ctrl+S) 108
Feature Type 108
Make Top Level Feature a Cut Out 108
Make Top Level Feature a Part 108
Import Tools 109
Import Silhouette, Centerline, or Color Image 109
Understanding silhouette and centerline images 109
Import Silhouette Image 110
Import Centerline Image 113
Import Color Image 115
Import DXF/DWG File 117
Create 118
Line 120
Rectangle 121
Arc 122
Circle 123
Text 124
Point 125
Polygon 125
Elliptic Arc 126
Ellipse 127
Spline 128
Shapes 130
Shape Tool 130
Pipe Joint Shape Tool (2D part) 131
Pipe Joint Shape Tool (pipe part) 132
Modify 133
Corner Tool 133
Extend Tool 134
Trim Tool 134
Scale Entities 135
Apply Offset 136
Fix Drawing 137
Curve Text 137
Cut Tool 140
Explode Entities 140
Group 141
Ungroup 141
Ungroup All 141
Ungroup vs. Ungroup All 141
Transform 142
Copy Entities 142
Delete Selected Objects 142
Move Entities 142
Rotate Entities 143
Measure Entities with Ruler 143
Bridge Entities 145
Perform Boolean Operation on Entities 146
Weld Selected Entities 148
Arc Fitting Tool 148
Mirror Entities 149

Linear Pattern Tool 150
Circular Pattern Tool 151
MANIPULATION TOOLS 151
Control handle grid *152*
Horizontal and vertical dimensions *152*
Rotation *152*
X and Y position *153*
LAYERS 153
Layer Controls *153*
Combining Layers *154*
VIEWPORT FOR CAD 155
Pan and Zoom Tools *156*
Pan 156
Zoom 156
Zoom Window 156
Zoom Extents 156
3D Orientation Tool *157*
Snap Tools *157*
All Snap 157
Center Snap 157
Endpoint Snap 158
Grid Snap 158
Horizontal/Vertical Snap 158
Intersection Snap 158
Midpoint Snap 159
Nearest Snap 159
Parallel Snap 159
Perpendicular Snap 160
Quadrant Snap 160
Tangent Snap 160

FLASHCUT CAM 161

FILE MENU FOR CAM 162
Printing 162
Print Nest Report 164
CAM RIBBON 165
File *165*
Feature Type *165*
Selection *166*
Settings *167*
Project Settings 167
Material Settings 168
Plasma Settings 170
Oxyfuel Settings 194
Waterjet Settings 196
Laser Settings 199
Marker Settings 200
Spindle Settings 201
Knife Settings 202
Hole Treatment Settings 204
Change Project Break Settings 205
Change Project Tab Settings 207
Manage Remnants 209
CAM Actions *210*
Grid Nesting 210
TrueShape Nesting 212
Common Line Cutting 215
Free Path 216
Bridge Objects 216

Layout (Pattern Nesting) 217
Multi Copy 218
Nesting Information 218
Fit workpiece to the parts 218
Connections tool 219
Simulation Tool 221
CAM Reset 221
Delete 221
G-Code 222
Create G-Code File 222
Open Editor 222
Import 222
DXF/DWG 222
CADCAM 222
VIEWPORT FOR CAM 223
Pan and Zoom Tools 223
Pan 223
Zoom 223
Zoom Window 223
Zoom Workpiece Extents 223
Zoom Toolpath Extents 223
3D Orientation Tool 223
Display Options 224
Cutting Directions 224
Kerf 224
Material 224
Parts 224
Part Numbers 224
Rapid Moves 224
Tool Paths 225
MANIPULATION TOOLS 225
Feature Manipulation 225
Lead In/Out Settings 225
Editing Settings 225
Manual Editing 226
Program Zero 226

FLASHCUT CNC 227

HARDWARE CONTROLS AND OTHER SETTINGS 228
Connect / Disconnect to Controller 228
System Status 228
General 228
Height Control Data 229
Fabrication Head Diagnostics 229
Reset Motor Drivers 229
Tool Rack Positions 229
Trace 230
Simulation 230
Toggle Display Units 230
Workpiece Information 230
FABRICATION HEAD CONTROL 231
Laser Pointer Controls 231
Marker Controls 231
Laser Controls 233
Plasma Controls 234
Oxyfuel Controls 235
Waterjet Controls 237
Spindle Controls 238

Hotwire Controls ... 239
Tangential Knife Controls ... 239
PROGRAM COORDINATES PANEL ... 240
Coordinate Rotation Example ... 241
Enter Rotation Angle ... 241
Use 2-Point Alignment ... 241
MACHINE COORDINATES PANEL ... 243
JOG / MOVE TO POINT / RIP CUT CONTROL PANEL ... 244
Jog ... 244
Axis Jog Buttons ... 244
Jog Mode ... 244
Jog Rate ... 244
Move to Point ... 245
Move Rate ... 245
Rip Cut ... 246
G-CODE CONTROL ... 246
Open G-Code file ... 247
File name ... 247
Load Multiple G-Code Files ... 247
Barcode ... 247
Run CAD Import Wizard ... 247
Reset G-Code ... 247
Jump to Line ... 247
Close G-Code file ... 247
Edit G-Code ... 248
G-Code Program Listing Box ... 248
Feedrate Override Controls ... 248
Run G-Code in Reverse ... 248
Feed Hold ... 248
Toggle G-Code Run Modes ... 248
Run G-Code ... 248
Progress meter ... 249
VIEWPORT FOR CNC ... 250
Multiple Views for 3D Operations ... 251
Lift Axis View ... 251
Viewport ... 251
Simulation View ... 251
3D Orientation Tool ... 251
Pan and Zoom Tools ... 251
Pan ... 252
Zoom ... 252
Zoom Window ... 252
Zoom Extents ... 252
Display Options ... 252
Material ... 252
Machine Envelope ... 252
Part Numbers ... 252
Show/Hide Grid ... 252

HELP ... 253

SYSTEM INFO ... 253
SUPPORT CONTACT ... 253
USER'S GUIDE ... 253
PROGRAMMING REFERENCE ... 254
BUILD SUPPORT FILE ... 254
Steps Recorder ... 254
SOFTWARE LICENSE AGREEMENT ... 254

Notices

Revised May 18, 2023
Version 10.1.6.10
© 1997-2023 WPI, Inc. All rights reserved.

FlashCut® CNC is a registered trademark of WPI, Inc.
Stingray® is a registered trademark of WPI, Inc.
PowerMax® is a registered trademark of Hypertherm, Inc.
AutoCAD® is a registered trademark of Autodesk, Inc.

The products and company names listed herein are trademarks or registered trademarks of their respective owners, and the use of the marks does not and should not imply any affiliation with, endorsement by, or authorization from, those companies.

About this document

Because new features may be added to the software in the future, this manual applies only to the version for which it was released.

There may be differences between the information presented here and the features found in either older or newer versions of the software.

Screen captures used in this document are edited for clarity and may differ in minor ways from the actual FlashCut application.

Disclaimer

FlashCut CNC and its affiliates are not responsible for the safe installation and use of this product. You and only you are responsible for the safety of yourself and others during the operation of your CNC machine tool. FlashCut CNC supplies this product but has no control over how it is installed or used. Always be careful!

FlashCut CNC is not responsible for damage to any equipment or workpiece resulting from use of this product.

If you do not understand and agree with all of the above, please do not use this product.

Safety and usage guidelines

Automated machining is potentially dangerous. Please take the time to completely read through this manual to understand operation of the software before running the system.

A working knowledge of the PC and the Windows operating system is required in order to install, use, and troubleshoot the software.

Since FlashCut is a real time control program, it must have full control of the operating system while running. It is very important that you do the following before running FlashCut:

- Disable all screen savers and power management programs.
- Make sure there are no background programs running, such as back-up software and calendar reminders.
- Make sure no other programs are open.

Safety is of the utmost importance. To use FlashCut CNC to control your automated machine tool in a safe and proper fashion, the following safety guidelines must be followed:

- Never let the machine tool run unattended.
- Require any person in the same room as a running machine tool to wear safety goggles, and to stay a safe distance from the machine.
- Allow only trained operators to run the machine tool. Any operator must have:
  - Knowledge of machine tool operation
  - Knowledge of personal computer operation
  - Knowledge of Microsoft Windows
  - Good common sense
- Place safety guards around the machine to prevent injury from flying objects. It is highly recommended that you build a safety shield around the entire tool envelope.
- Never place any part of your body within the tool envelope while the machine has power, since unexpected machine movement can occur at any time.
- Always keep the tool envelope tidy and free of any loose objects.
- Be alert for computer crashes at all times.

Technical support

Expert technical support is provided for all our products. Many resources are available to help you resolve your problems quickly. We recommend that you use these resources in the following order:

Website

http://www.flashcutcnc.com/

Our website has product specifications, documentation, support videos, and other information.

Dealer support

If you purchased FlashCut CNC from a dealer or other machine tool manufacturer (OEM), please contact them first. They will have the best knowledge of your complete system.

Email

support@flashcutcnc.com

Email is the most organized way to communicate an issue to our support staff. In your e-mail, please state your problem completely. The email should include this information:

- FlashCut version
- Computer processor and speed
- Windows version
- Signal generator serial number.

In addition, please attach the following files:

- Setup and Tooling files (usually found in a folder named **c:\FlashCut Data**)

- G-Code file with which you are having problems (when appropriate)
- CADCAM file with which you are having problems (when appropriate)

Alternatively, you can attach a single FlashCut support file. The support file is in ZIP format and contains all relevant files needed by technical support to resolve your issue. To generate the file:

- Select the Help button in the upper right corner of the main window.
- Click the Build Support File button.
- Name and save the file using the Windows dialog box.

Refer to the Help section of this document for more details.

Phone/fax support

If email is unavailable to you, please call our telephone support number. We will normally respond to your call within 24 hours.

Phone: (847) 940-9305 (9:00 AM-5:00 PM, US Central Time, M-F)
Fax: (847) 940-9315

Introducing FlashCut® CNC

Thank you for purchasing FlashCut® CNC, the most innovative, powerful CNC control system for Windows 10 and 11. No control system is easier to set up and use. With intuitive controls and real time graphics, FlashCut CNC lets you design and make parts quickly and accurately on your machine tool.

FlashCut has three main portions – FlashCut CAD, FlashCut CAM, and FlashCut CNC.

FlashCut CAD – A simple and powerful 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 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.

System Requirements

Windows 10 or higher (64 Bit)

Intel i3 processor (or equivalent), or better

2.1 GHz clock speed or greater

8GB of RAM or more

A separate graphics card is preferred for processing larger files

Installing the FlashCut CNC Software

Preparing to Install the Software

Important: Install FlashCut CNC software before connecting the USB cable between the computer and the controller (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.

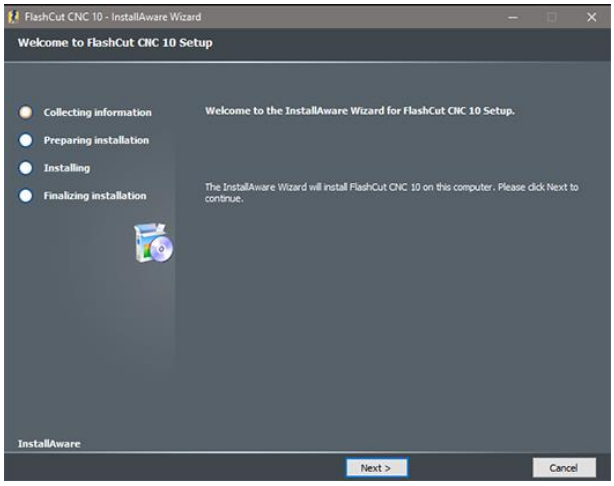
When the hardware is connected to the computer with the USB cable and then powered on, Windows will try and find a driver for the device. If the correct driver is not already installed, Windows may attempt to use a driver for a different device. If the software is installed first, the proper driver will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

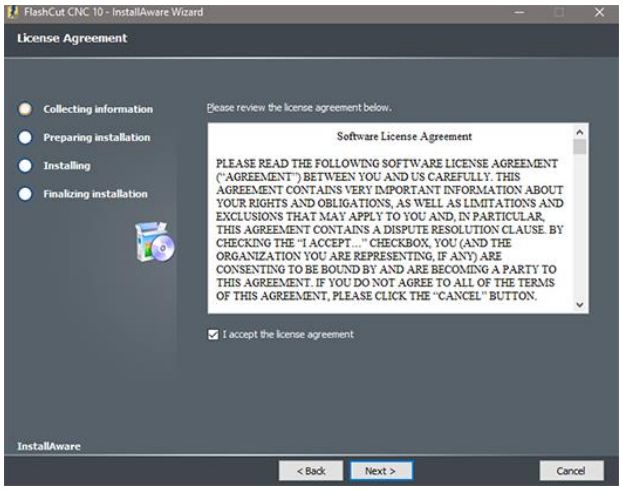
Installing the Software

Please follow these instructions to install FlashCut CNC.

1. Download the FlashCut CNC installer file from the internet or from a USB drive.
2. Run/Open the installer program. This will launch the InstallAware Wizard.

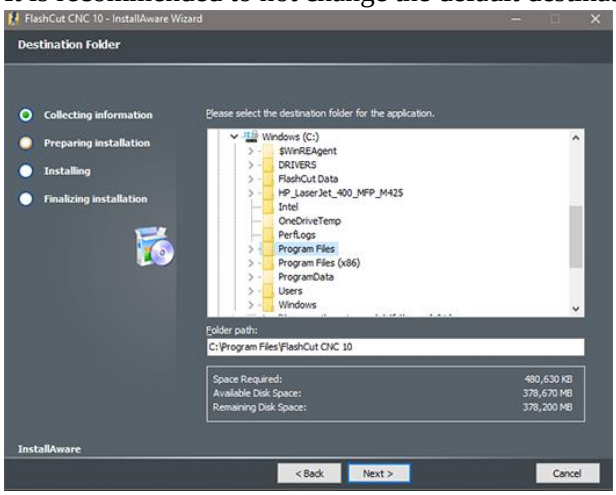
3. After verifying pre-installation requirements, the wizard prompts you to install. **Next >**

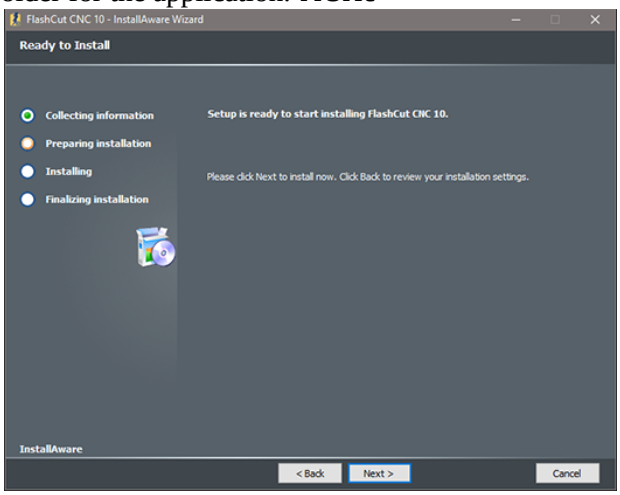




4. Review the software license agreement. **Next >**

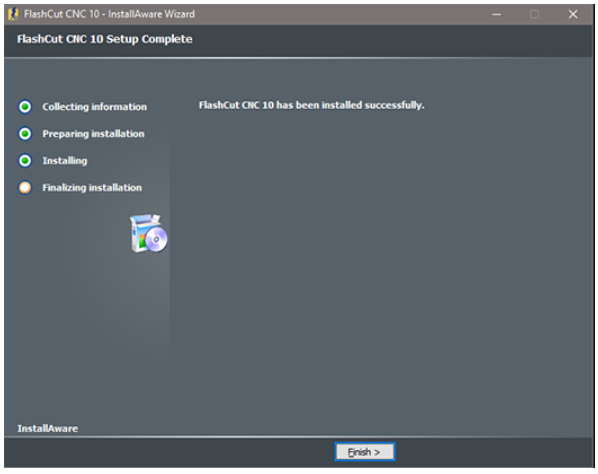
5. It is recommended to not change the default destination folder for the application. **Next >**





6. The wizard is now ready to perform the installation and configuration. **Next >**

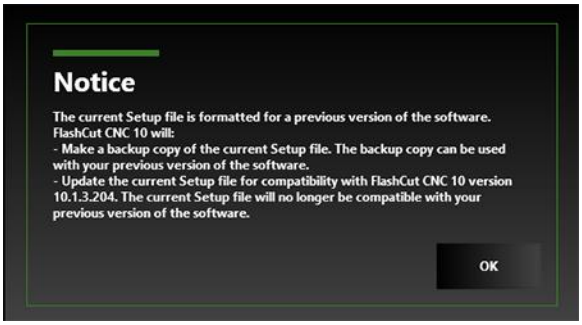
During the installation process, Windows may display this message asking for permission to install the USB Signal Generator 501A driver. Click **Install** to install the driver software.



7. When complete, the wizard will inform you if the installation was successful. **Finish >**

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, refer to the **Configuring FlashCut** section of this document.



Installing the USB Driver

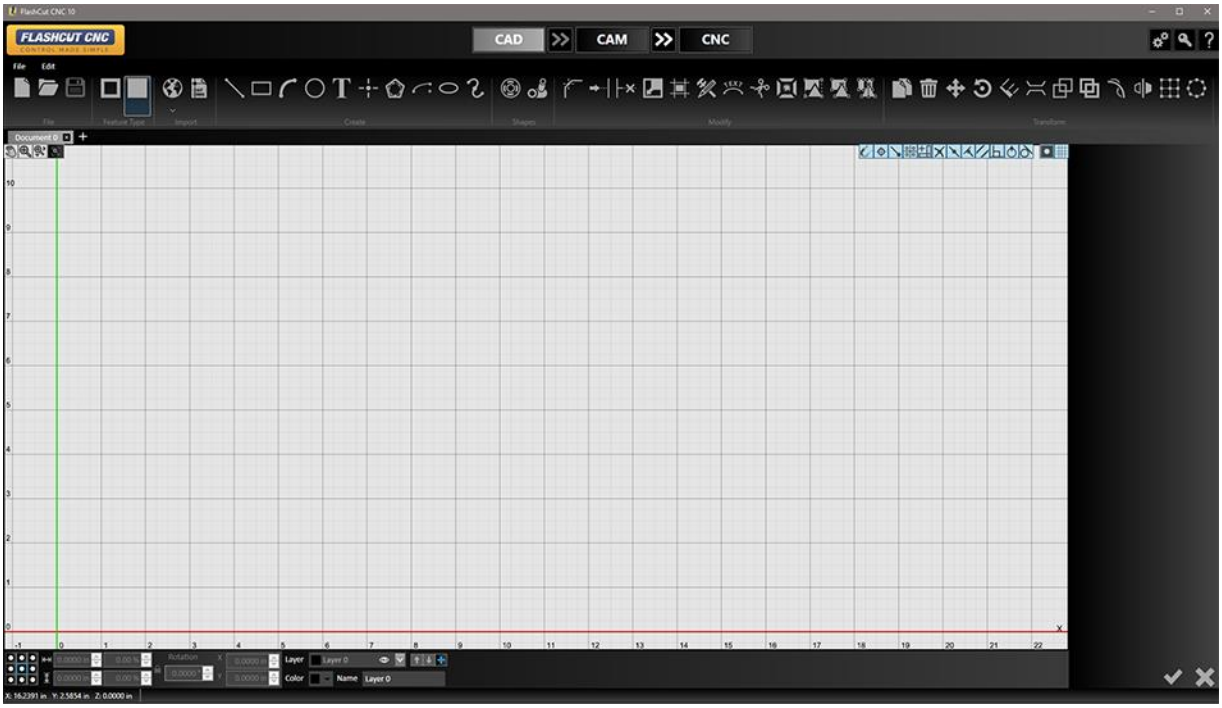
The software needs to be installed before you connect a signal generator to the PC. If you connect the electronic hardware before installing the software, there is a high likelihood that you will have to manually install the USB driver. The USB driver will be automatically installed if the FlashCut CNC 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.

Getting Started



Navigating Through CAD CAM CNC

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

Use the **Double Arrow** buttons 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.

Apply / Cancel

The **Apply** and **Cancel** buttons are used throughout CAD, CAM, CNC, including some dialog windows.

The green check is used to **Apply** an action or operation, and the red X is used to **Cancel** an action or operation.

Running in Evaluation Mode

After you install FlashCut CNC it will run in "Evaluation Mode" until you activate a license. In evaluation mode, you can try most 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 a limited number of lines of G-Code will be generated by CAM.

Licensing FlashCut

Click the **License** button to display the FlashCut license management window.

There are two forms of licensing in FlashCut: a Controller License that is a *.clu file for the controller and a PC License that is a license key for a computer.

Several features within the software are enabled (or disabled) via the software License. The features enabled by the currently installed license can be reviewed in this window.

Controller License

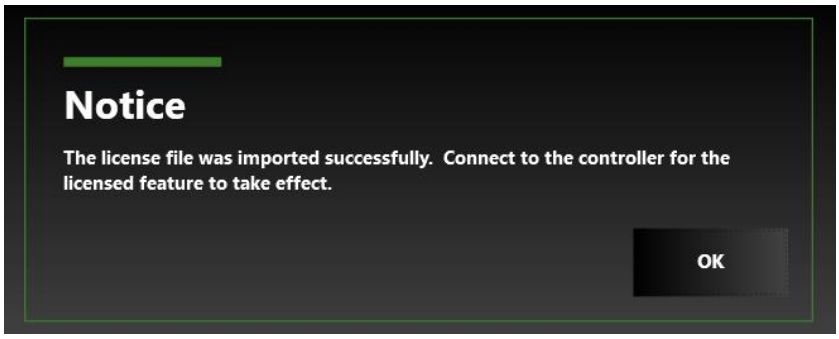
Typically, when you purchase a new control system from FlashCut, your controller will already be licensed. In that case, no additional action is necessary; you can simply Connect to the controller and use the software.

However, if you are upgrading your system and have received a new/updated license, you will need to import the new controller license update file for the software license and features to take effect. The controller license file has a .clu file extension and the file name includes the serial number of your controller and version of software. For example: LicenseUpdate_V10_S05123410123_1.clu

Import License File

Follow these steps to import the controller license file:

1. When you receive a controller license update file, download a copy of it to a location on your computer, such as the Desktop.
2. Click the **Import License File...** button.
3. In the dialog window, navigate to the location of the file and press the **Open** button.
4. A message will pop up to confirm the file was imported. Press **OK**. But you're not done yet!
5. Go to CNC and press the Connect button. Upon a successful Connection to the controller, the updated license should now be in effect. Review the list of license features to confirm.



PC License

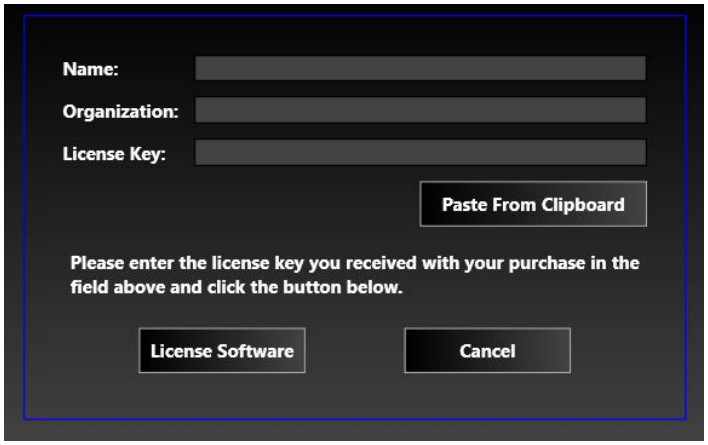
If this is the computer at the fabricating machine and you are Connecting to a licensed controller, it is not generally necessary to also activate a PC License on that same computer. A typical use of the PC License is to use the software on an office computer or on a laptop computer that is not always connected to the controller hardware.

Note When activating (or deactivating) a PC License, an internet connection is required.

Activate PC License...

Follow these steps to activate a license key:

1. Click the **Activate PC License...** button.
2. A dialog box opens.



3. Enter the Name and Organization (company name). If you've copied the license key from an email you received, you can use the **Paste From Clipboard** button, or you can type the license key in the License Key field.
4. Click the **License Software** button.
5. FlashCut installs the license and displays a notice to confirm that the license has been activated.

Deactivate PC License

You may deactivate the license on your computer so it can be used on a different computer. Follow these steps to deactivate the license on your computer.

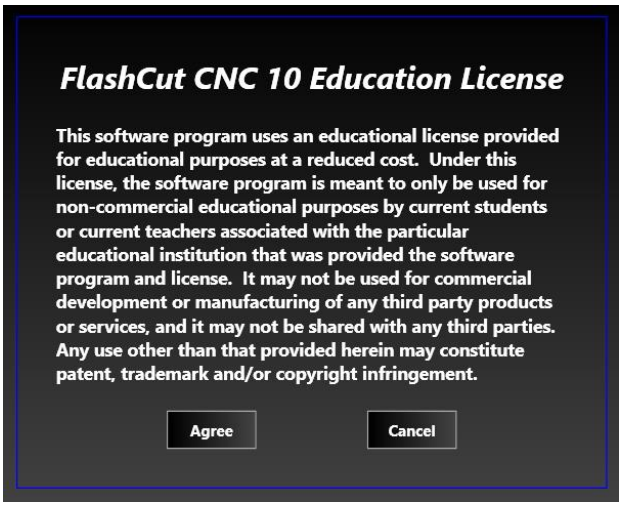
1. Click the **Deactivate PC License...** button.
2. FlashCut displays a confirmation dialog box.
3. Click **OK** to deactivate the license.

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 all available features, refer to the FlashCut CNC website or contact a sales representative.

Demonstration License

Some licenses are intended only for the purpose of demonstrating and selling this product. If this is not how you are using this product, contact FlashCut technical support.

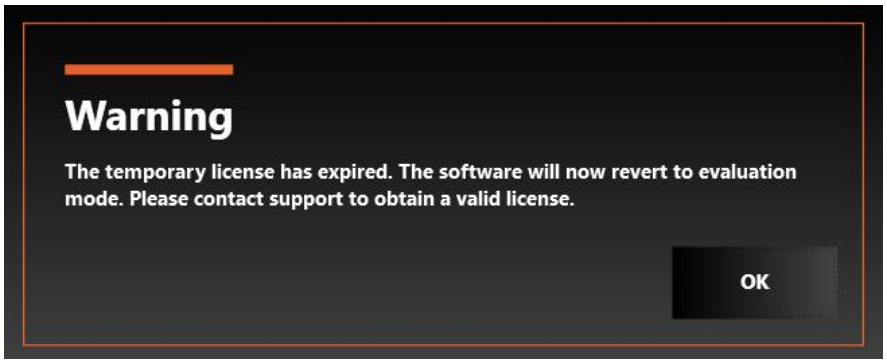


Educational License

Some licenses are intended only for educational purposes. If this is not how you are using this product, contact FlashCut technical support.

Temporary License

Some licenses are temporary licenses that are set to expire. If your license has expired, contact technical support to get an updated license.

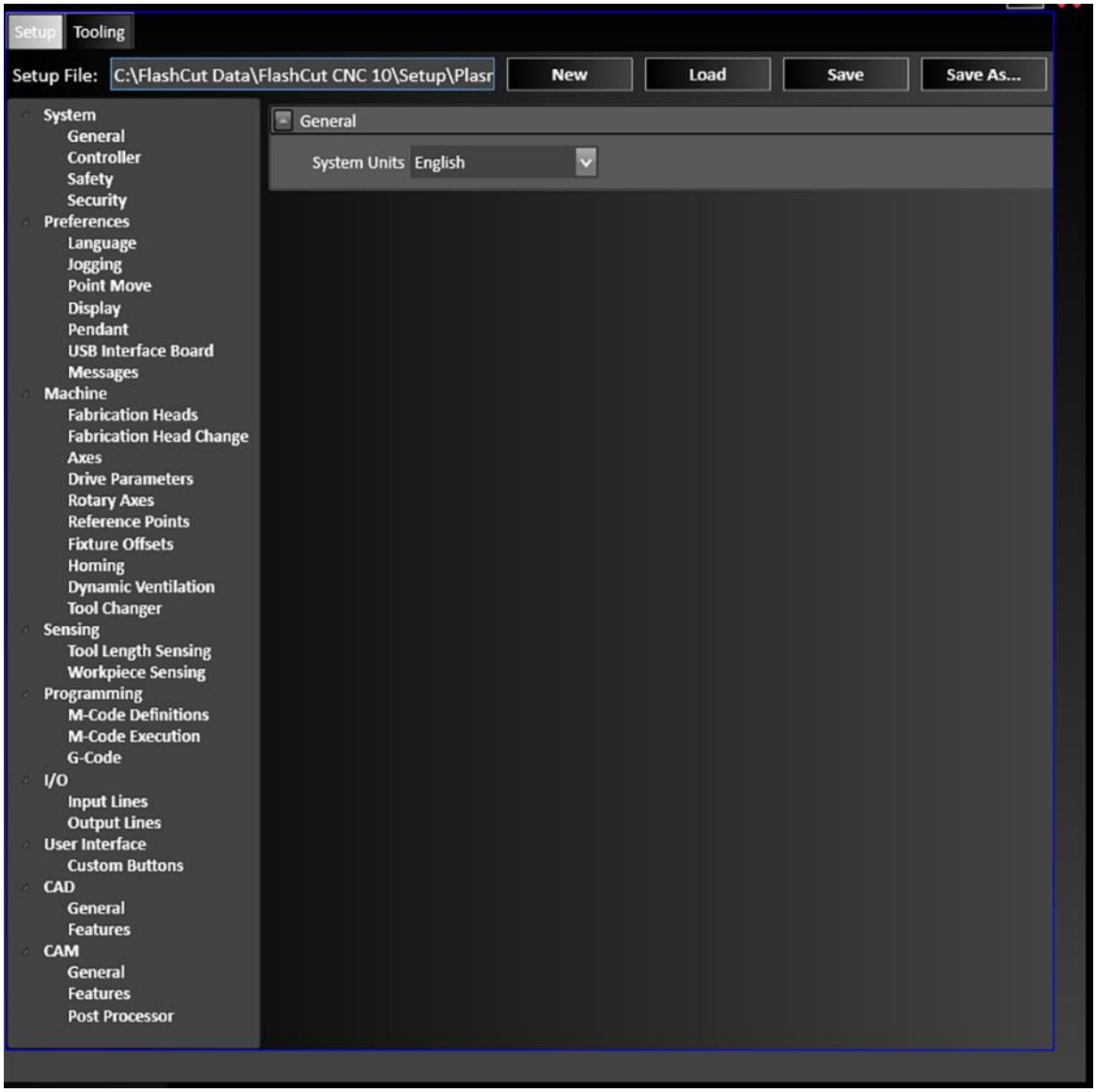


Configuration

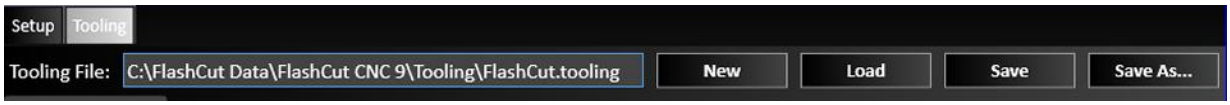
This section describes how to configure FlashCut for use with your machine.

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. Refer to the Security section below for more information.



Configuration Files



Setup File Selection

The Setup file contains all of the configuration parameters for your machine.

When FlashCut is first launched, it may prompt you to select a setup file, or it may simply load a default file. If you were upgrading from a previous version of FlashCut, the previous setup file will be updated and loaded.

The default location FlashCut stores setup files is in the **C:\FlashCut Data\FlashCut CNC 10\Setup** folder.

The **New** button allows you to load a pre-packaged setup file. Select the file then click **OK**. A copy of that file will be loaded. If your system has multiple pre-packaged setup files, use this to select the setup file that is configured for your system.

The **Load** button allows you to load a different setup file. Setup files have the extension ***.setup**. Older, legacy files with the extension *.stp, from Version 4.5 and newer, can be loaded and will get updated.

Click **Save** to save the current configuration state and close the Configuration dialog.

Use **Save As...** to save a copy of the setup file. You can choose a different name or destination for the file.

Tooling File Selection

The Tooling file contains the tool library parameters for your machine tools.

When FlashCut is first launched, it may prompt you to select a tooling file, or it may simply load a default file. If you were upgrading from a previous version of FlashCut, the previous tooling file will be updated and loaded.

The default location FlashCut stores tooling files in the **C:\FlashCut Data\FlashCut CNC 10\Tooling** folder.

The **New** button allows you to load pre-packaged tooling file. Select the file then click **OK**. A copy of that file will be loaded. If your system has multiple pre-packaged tooling files, use this to select the tooling file that is configured for your system.

The **Load** button allows you to load a different tooling file. Tooling files have the extension ***.tooling**.

Click **Save** to save the current configuration state and close the Configuration dialog.

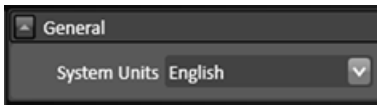
Use **Save As...** to save a copy of the tooling file. You can choose a different name or destination for the file.

Setup Configuration Menu

System

General

General

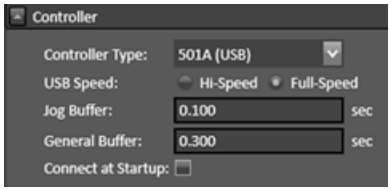


System Units

Select English or Metric. This defines the units used for most machine configuraton settings and the default units when interpreting G-Code programs.

Controller

Controller



Controller Type

Specifies the type of signal generator (controller) you are using: 501A (USB)

USB Speed

Allows you to specify the speed of USB communications with the signal generator. Setting the USB speed to **Full Speed** is recommended to help prevent communication problems.

Jog Buffer

The jog generator buffer prevents system 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 movement 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.

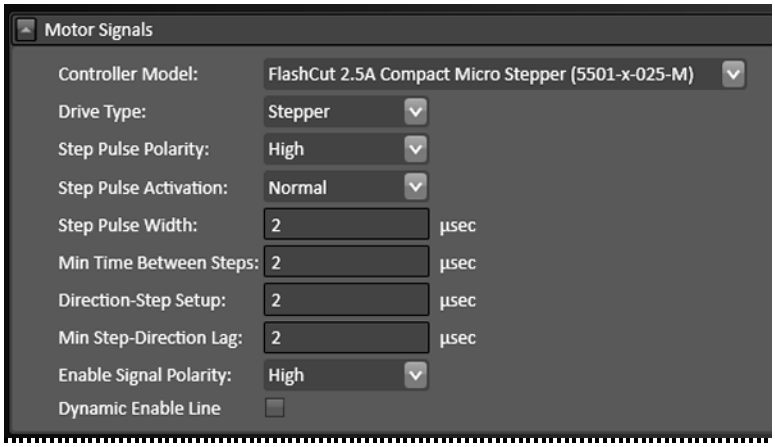
General Buffer

Sets the buffer size on the signal generator for all operations except jogging in continuous mode.

Connect at Startup

When this option is selected, the application will Connect to the controller when the application launches. Note: A user will need to acknowledge the Safety message.

Motor Signals



Controller Model

Sets 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).

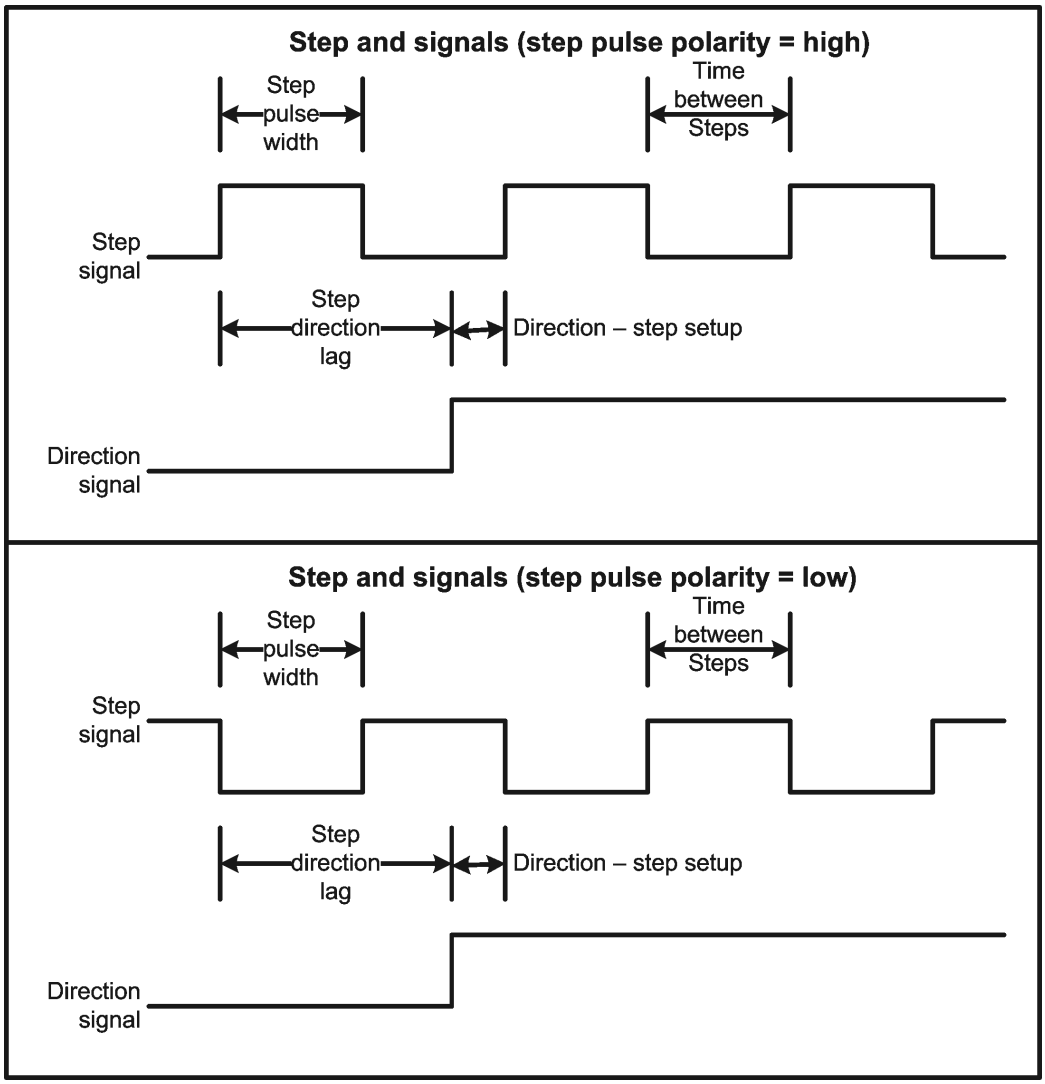
If you are using non-FlashCut drivers, choose **Other**. This enables all fields, allowing you to enter the proper settings based on your motor driver requirements. Note that some signal fields do not apply to all signal generator models and may be disabled based on your controller settings (signal generator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

Drive Type

Indicates your motor driver type: stepper or servo.

Step Pulse Polarity

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



Step Pulse Width

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

Min Time Between Steps (µsec)

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

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.

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.

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.

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.

Signal State Preferences

Enable Motors On Power Up

Sets the state to which the signal generator drives the motor enable line when turned on, to initially enable or disable the motors.

Enable Motors After Disconnect From Sig. Gen.

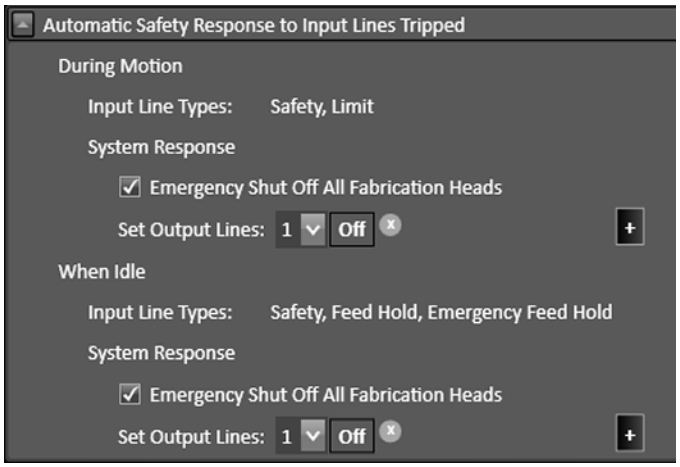
Sets the state to which the signal generator drives the motor enable line when you disconnect from the signal generator.

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.

Safety

Automatic Safety Response



During Motion

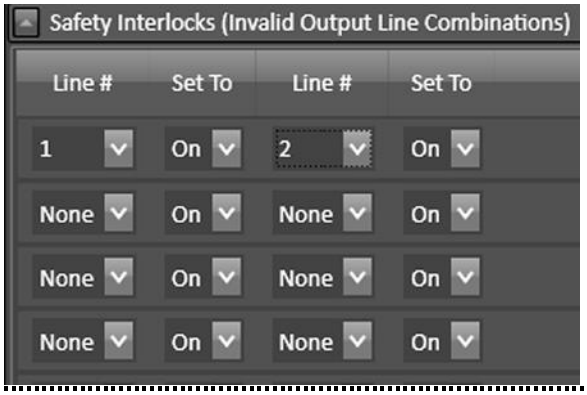
After a Safety or Limit Input Line becomes tripped, specify the desired System Response.

When Idle

After a Safety or Feed Hold Input Line becomes tripped, specify the desired System Response.

A typical use of this feature is to turn off the fabrication head when a safety enclosure door is opened.

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 pull-down menus across a row of the table to define the incompatible states of two 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 pull-down menus on the first line of the table would be set as shown:

Output Line #: 1

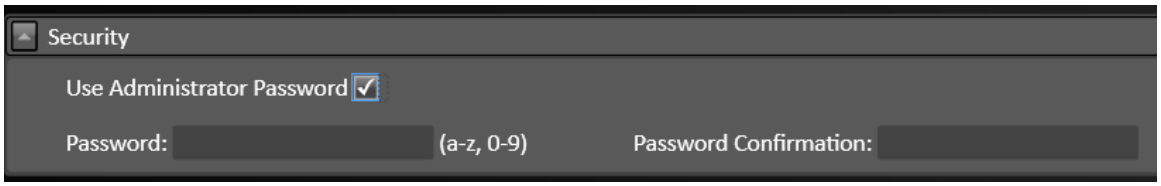
Set To: On

Output Line #: 2

Set To: On

Security

Security



Use Administrator Password

Check this box to set or change the Administrator Password. Enter the desired password in the Password and Password Confirmation fields and hit Save.

When an Administrator Password is in use, you must enter the password when you enter the Configuration interface.

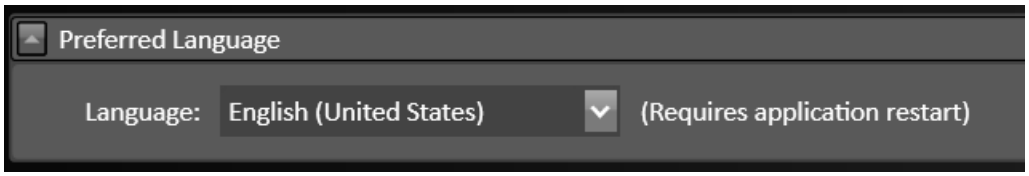
Password and Password Confirmation

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

Preferences

Language

Preferred Language



Choose the language FlashCut uses for its user interface. Note: Although several items may become translated immediately, an application restart is necessary for the language change to take effect throughout the application.

English (United States)

Spanish (Spain)

Italian (Italy)

German (Germany)

French (France)

Dutch (Netherlands)

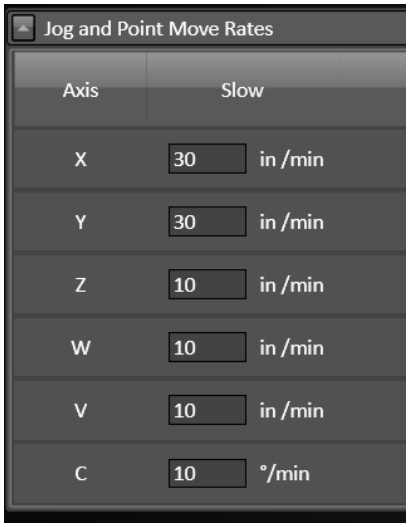
Portuguese (Portugal)

Japanese (Japan)

Jogging

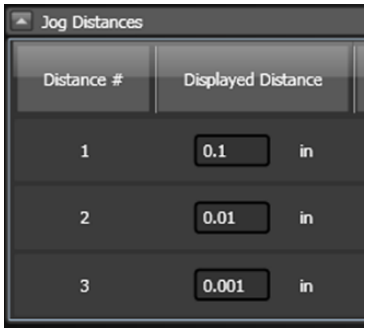
Jog and Point Move Rates

Specify the Slow feedrate for Jog and Point moves. There is a slider bar in the Jog and Point Move control panel in CNC that allows you to specify a feedrate range between "Slow" and "Fast". The "Fast" feedrate is the Maximum Feedrate that is specified for each axis in the Axes configuration.

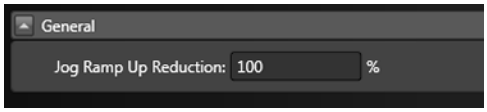


Jog Distances

In the Jog control panel in CNC, you can select Continuous, Step, or one of three discrete jog distances. Specify the three distances here.



General

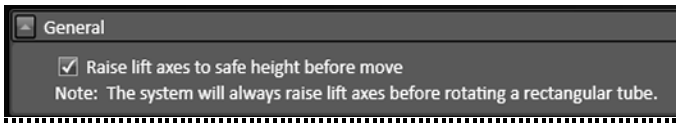


Jog Ramp Up Reduction

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

Point Move

General

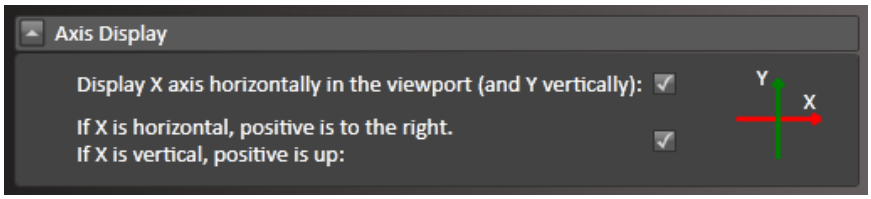


When selected, the default action will be to raise lift axes to safe height before more.

Note: The system will always raise lift axes before rotating a rectangular tube.

Display

Axis Display



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

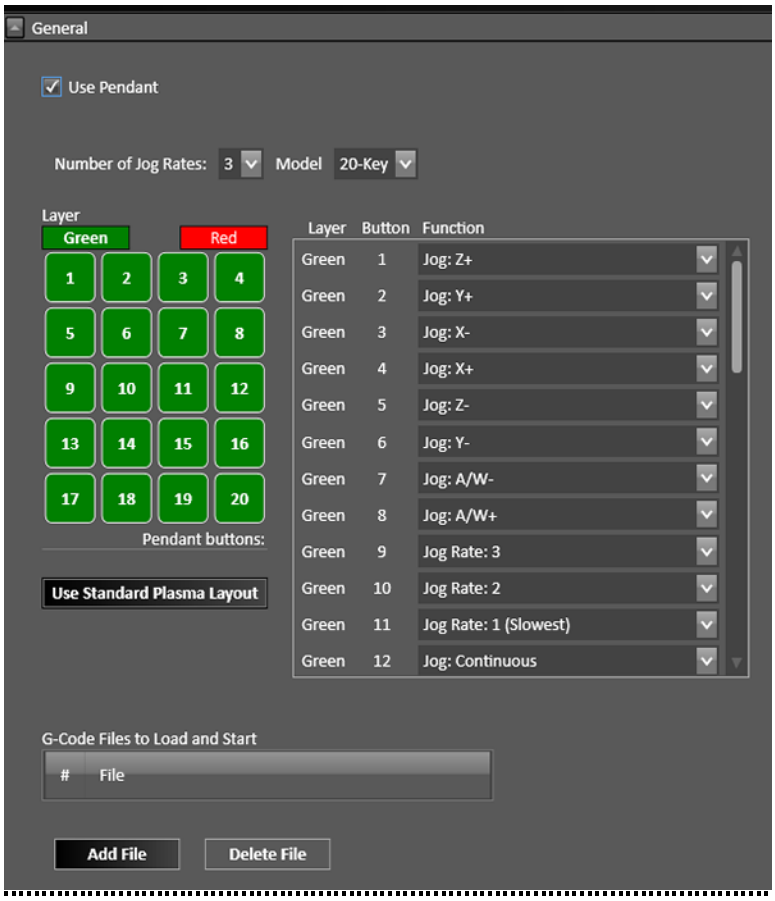
Current Position

Select the desired icon to mark the fabrication head position within the viewport.

Pendant

General

The pendant is a versatile accessory that you can customize to your application. Select the Model of pendant that you have: 20-key, 24-key, or 12-key with joystick. Then you can select a predefined standard layout for the key functions, or you can assign functions to each button to create your own custom layout.



G-Code Files to Load and Start

Up to 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.

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.

General

You can optionally specify a Device Name. Use the Add Function and Delete Function buttons to define or remove the functions assigned to different row, column, and layer combinations of the wiring.

Layer (green or red)

Wiring Row (1-8)

Wiring Column (1-12)

File Table

Up to 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.

Messages

In some scenarios, before starting a G-Code program, a warning or informational message will pop up.

Enable or disable those specific messages here.

Machine

Fabrication Heads

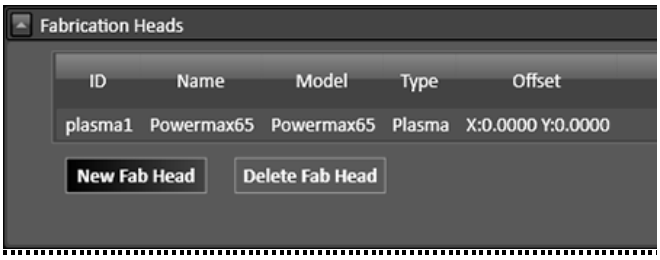
This section explains in detail how to configure each of the different types of Fabrication Heads used by FlashCut.

Fabrication Heads

Fabrication heads are the cutting or marking heads on your machine.

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

Plasma, Oxyfuel, Waterjet, Laser, Mill, Tangential Knife, Hotwire, Marker, and Laser Pointer.



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 a 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 where you will then be prompted to select the fabrication head **Type**. 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.

There are some **General** settings that are common for all fabrication heads:

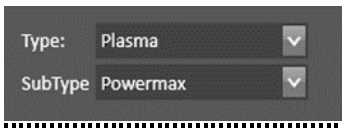
ID#: This is a number that is used with the fabrication head Type to create a unique identifier of the fabrication head.

Name: This defaults to the fabrication head Type (or Model). It can be edited, if desired.

Offset: When using more than one fabrication head, measure the X and Y offsets from the other fabrication head and enter the measured values.

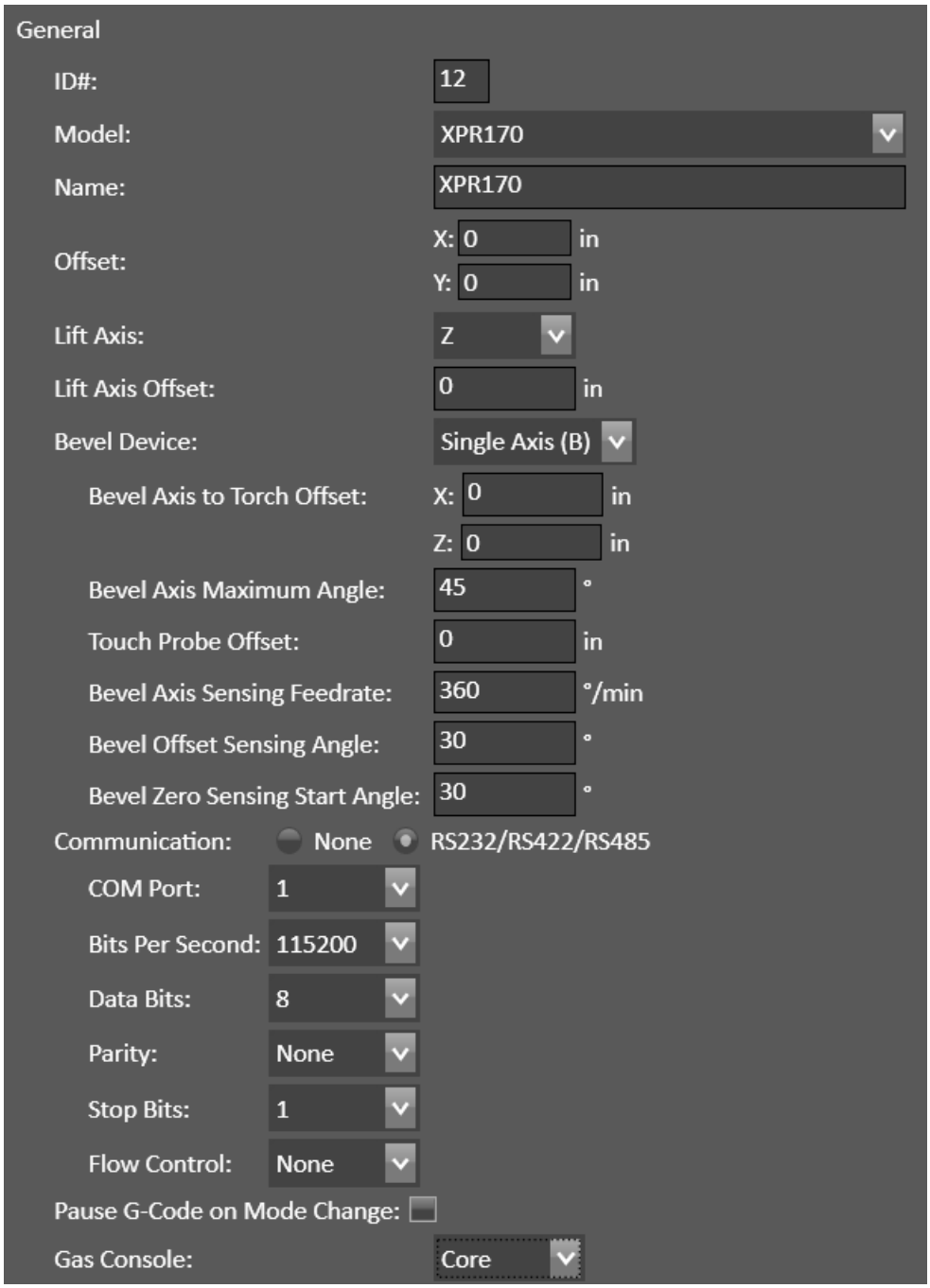
Plasma

When adding a **New Fab Head** where the **Type** is **Plasma**, you will also be prompted to select a **SubType**. Select from a list of the supported plasma model types. For example: Powermax, Cutmaster, etc…



General

The available parameters will vary depending on the plasma model and the features available on your system.



Model: For most types of plasma systems, you can select from a list of specific plasma models that are supported. For example: PowermaxSYNC65, PowermaxSYNC85, etc...

Lift Axis: This defaults to the Z axis. A different axis can be selected (W V U). A typical scenario is when multiple fabrication heads are configured, where you may also need to measure the **Lift Axis Offset** value. If your plasma system does not use a lift axis, select None. Several lift-axis-specific features will no longer be available when there is no lift axis.

Bevel Device: If your system is capable of beveling, this is where to define the bevel parameters for the fabrication head. At this time, the system only allows it to be the B axis.

Lead Length: For some Plasma models, such as Cutmaster, FlexCut, MaxPro, you specify the lead length. Choose the appropriate length from the menu.

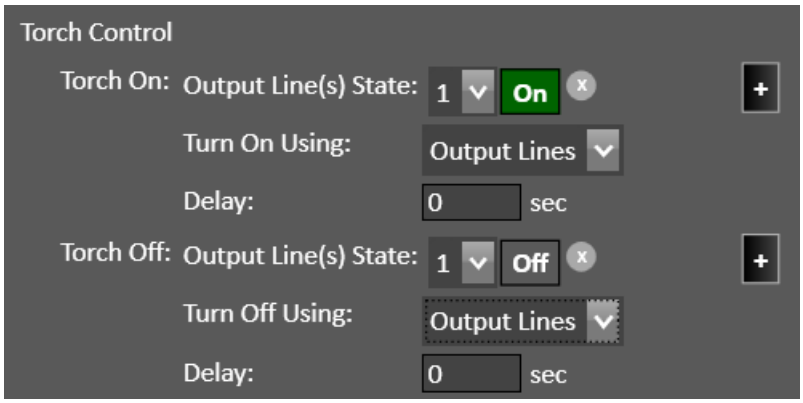
Communications: Some Plasma models support communications. Extra hardware is required to use this feature. Depending on the model, this may simply control the amperage or on more complex plasma systems this can control cutting gas mixes, status codes, and many other features.

Pause G-Code on Mode Change: If your plasma system is capable of both cutting and marking, this feature can be useful when you need to manually change consumables before using a different mode.

Torch Model / Gas Console: For some Plasma models you need to specify the torch model or gas console model. For example, the SmartFocus200 has the option of the PerCut2000 or PerCut4000. The **Post Flow Time** can also be specified.

Torch Control

It is required to specify the basic on/off controls for your plasma system.



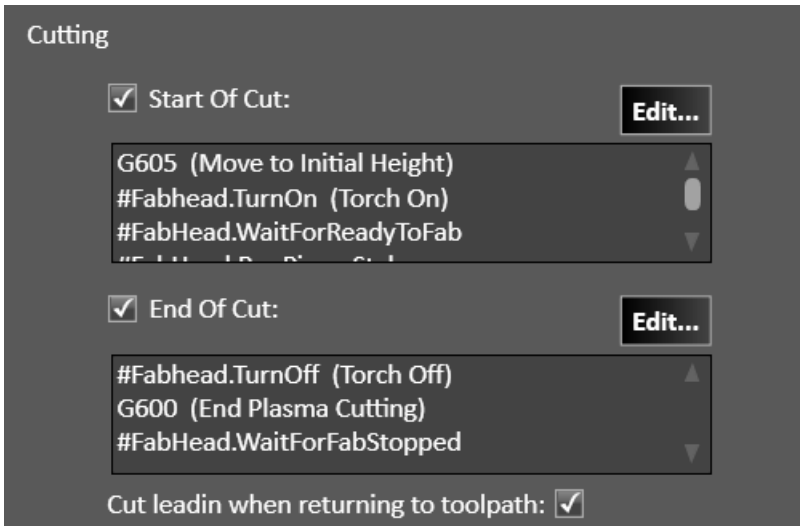
Torch On/Off Output Line(s) State: For most plasma systems, this will be Output Line #1 On for Torch On and Output Line #1 Off for Torch Off. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most plasma systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most plasma systems, no delay is required.

Cutting

Specify the commands to be executed at the Start and End of each cut in a G-Code program.



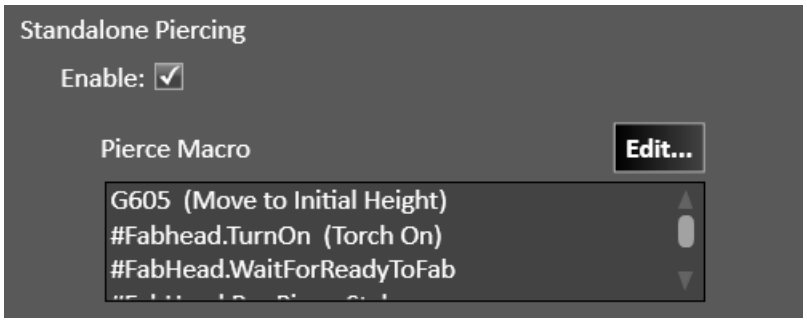
Start of Cut: For most plasma systems, the default set of commands in the macro will work well. These commands will be executed automatically when starting a cut in the G-Code program. Typically, this includes doing a Touch off, turning on the plasma torch, waiting for the arc transfer signal, and executing a pierce. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: For most plasma systems, the default set of commands in the macro will work well. These commands will be executed automatically when ending a cut in the G-Code program. Typically, this includes turning the torch off and waiting for a confirmation of loss of the arc transfer signal. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Cut leadin when returning to toolpath: Enable this feature if you want to cut a lead-in from the plasma torch's current position to the toolpath when resuming a G-Code program.

Standalone Piercing

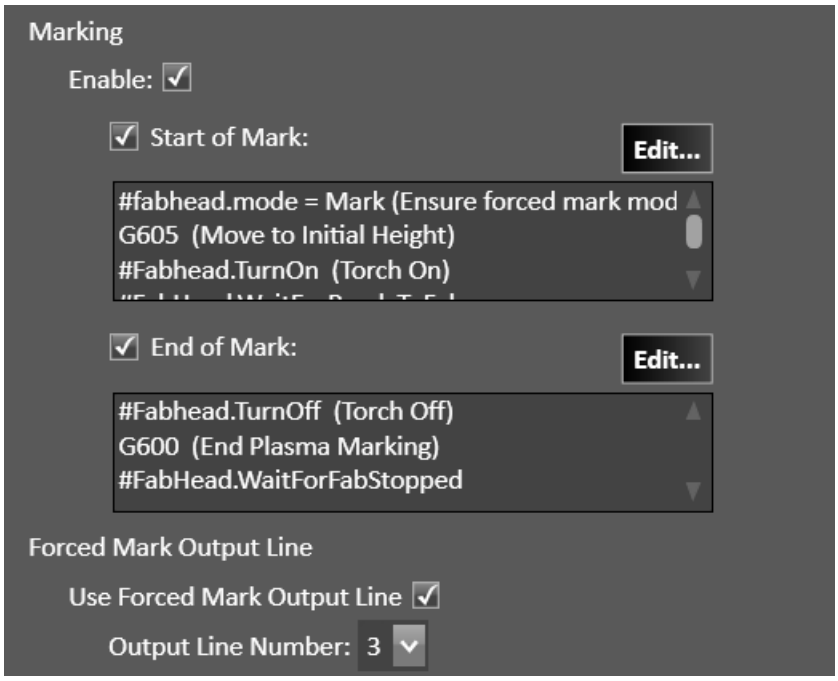
Specify the commands to be executed for a standalone (separate) pierce in a G-Code program.



Pierce Macro: For most plasma systems, the default set of commands in the macro will work well. These commands will be executed when a pierce command is in the G-Code program. Typically, this includes the same commands found in the Start of Cut macro and End of Cut macro. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Marking

Specify the commands to be executed at the Start and End of each mark in a G-Code program.



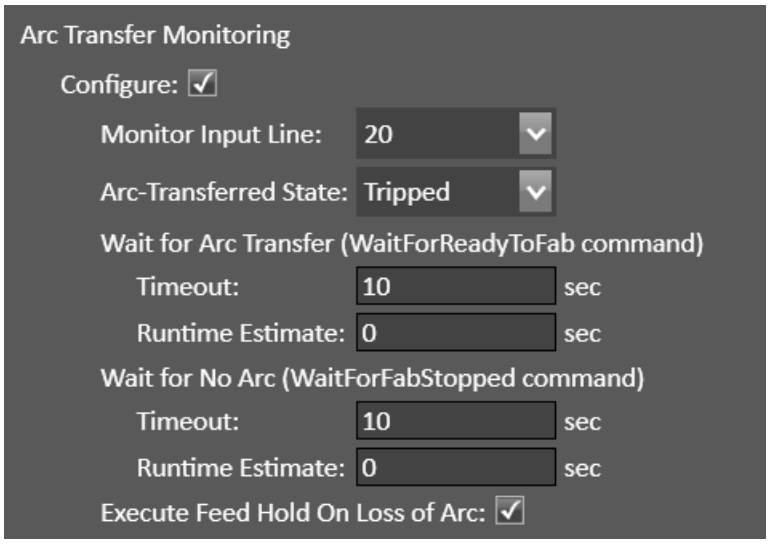
Start of Mark: For most plasma systems, the default set of commands in the macro will work well. These commands will be executed automatically when starting a mark in the G-Code program. Typically, this includes doing a Touch off, turning on the plasma torch, and waiting for the arc transfer signal. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Mark: For most plasma systems, the default set of commands in the macro will work well. These commands will be executed automatically when ending a mark in the G-Code program. Typically, this includes turning the torch off and waiting for a confirmation of loss of the arc transfer signal. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Forced Mark Output Line: Some plasma models, such as the Lincoln FlexCut125, use a signal to put the system into marking mode.

Arc Transfer Monitoring

This defines what input line to monitor for the arc transfer signal. This information is used by the commands #FabHead.WaitForReadyToFab and #FabHead.WaitForFabStopped typically used in the macros for Start of Cut and End of Cut.



Monitor Input Line: The input line for the arc transfer signal. For most plasma systems this will be 20.

Arc-Transfer State: The state of the arc transfer signal input line: Tripped or Normal. For most plasma systems this will be "Tripped".

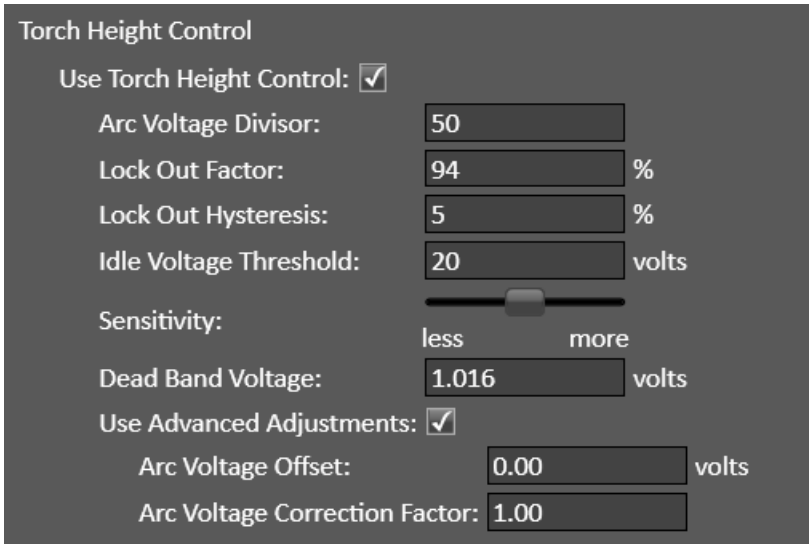
Timeout: This is the timeout duration, before a message pops up, to wait for the arc transfer signal state.

Runtime Estimate: The actual time it takes for the system to wait for the arc transfer signal can vary. This value is used to help calculate the run time estimate that is used for the progress meter while running a G-Code program in CNC and for the nesting information report in CAM.

Execute Feed Hold On Loss of Arc: While running a G-Code program, this will Feed Hold the program if the system detects a loss of the arc transfer signal.

Torch Height Control

Specify the parameters used for automatic height control of the lift axis.



Arc Voltage Divisor: For most plasma systems there is a 50:1 voltage divisor. Verify what the divisor is for your specific torch. For example, the HyPerformance models of plasma torches typically have a 42:1 voltage divisor.

Lock Out Factor: Temporarily suspend height control when the machine slows below this percentage of the normal feedrate. This helps to prevent diving down into the material; for example, when the machine slows down while cutting a corner. A typical value is 94%.

Lock Out Hysteresis - The height control will resume when the machine is back at the normal feedrate. However, to help prevent the height control system form being suspended too frequently, this hysteresis formular is used: ((Lock Out Factor) + (Lock Out Hysteresis))* (Programmed Feedrate). A typical value is 5%.

Idle Voltage Threshold: This is just for aesthetics. When the torch is idle, the actual displayed voltage is very low and may bounce around by several volts. To avoid this, no voltage will be displayed that is below this value. Typically set to 20 volts.

Sensitivity: Determines how sensitive the height control reacts towards voltage fluctuation. The slider control corresponds to the **Dead Band Voltage** displayed below. Most plasma systems work well when the dead band voltage is about 1 volt. This can be adjusted to be more or less sensitive based on your plasma system and the type of material that is being cut.

Use Advanced Adjustments: Most plasma systems do not need to use this. However, some systems may need configurable voltage adjustments to correct for inaccuracy of the measured arc voltage that may be caused by the plasma torch lead length and other factors. The **Arc Voltage Offset** and **Arc Voltage Correction Factor** can be used to help correct these inaccuracies.

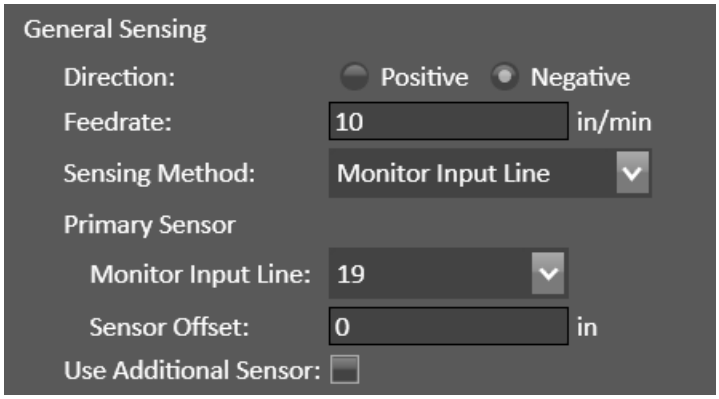
WARNING: Adjusting these fields outside of the recommended procedure can produce undesired effects. It is important that the calibration is used to accurately measure incoming voltage and not to compensate for outdated cut charts, poor gas quality, incorrect voltage divisors values, or other common voltage problems.

1. IMPORTANT: Disconnect the DB-15 plug from the THC Isolation Box before proceeding. Failure to do so will result in an inaccurate calibration and may DAMAGE both the FlashCut CNC control system and the plasma power supply.
2. Open the software configuration and navigate to the “Fabrication Heads” page. Click on the plasma in the list to drop-down additional settings. Then, scroll to the Torch Height Control section.
3. Determine the Arc Voltage Divisor value per the plasma manufacturer's documentation and make sure that this value matches what is entered in the software.
4. Ensure that the “Use Advanced Adjustments” checkbox is unchecked.
5. Save the Configuration and return the CNC module.
6. Using the manual Torch On/Off button in CNC, turn the torch output ON (lit up blue).
7. Remove the lid of the isolation box (attached by the four Phillips screws on top face of lid).
8. To perform basic Offset calibration:
    a. Use Jumper JP200 to inject a voltage. Take reading from CNC screen THC section.
    b. Jump to 2V. Record V2 _____V Optional: Measure injected voltage record here 2v Actual _____V.
    c. Jump to 3V. Record V3 _____V Optional: Measure injected voltage record here 3v Actual _____V.
    d. Calculate Arc Voltage Correction Factor: Correction Factor = (3vActual - 2vActual)*ArcVoltageDivisor/(V3-V2) = _____
    e. Calculate Offset: Offset = V2-CorrectionFactor*2vActual*ArcVoltageDivisor
9. Save the **Arc Voltage Offset** and **Arc Voltage Correction Factor** in the Configuration.

10. Check your work: 2V jumper should read near 100V for 50:1 divisor, 3V jumper should read near 150V for 50:1 divisor.

General Sensing

Specify the sensing parameters to be used for Touch Off and Program Zero Sensing of the lift axis.



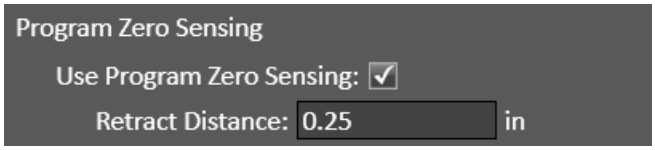
Direction: This is typically Negative, down toward the material.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 10 in/min. Setting this value too high can result in inaccurate results or other errors.

Sensing Method: This can be "Monitor Motor Current" if the lift axis uses a servo drive. You specify the **Current Limit** and a **Current Debounce**. Or this can be "Monitor Input Line" and you can use a Primary Sensor and optionally **Use Additional Sensor**. You specify the **Monitor Input Line** and **Sensor Offset**. A typical use is to have the Primary Sensor be an ohmic sensor, which is typically Monitor Input Line 19 with 0 Sensor Offset.

Program Zero Sensing

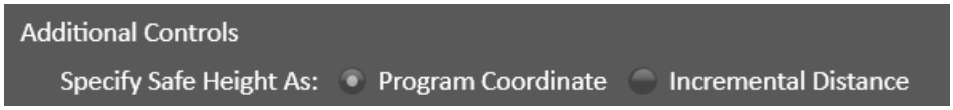
This will enable program zero sensing in the Program coordinate's Set menu.



Retract Distance: This is the distance the lift axis will retract after sensing program zero.

Additional Controls

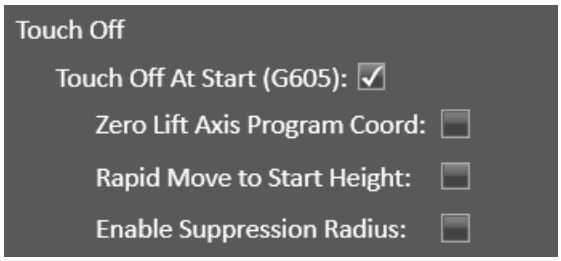
The only additional controls are how to specify the Safe Height coordinates.



Specify Safe Height As: This is typically set to "Program Coordinate". For some systems "Incremental Distance" may be preferred.

Touch Off

Enable the use of Touch Off (the G605 command). Controls will be displayed in CNC. If your system has advanced touch off features, those can be enabled here.



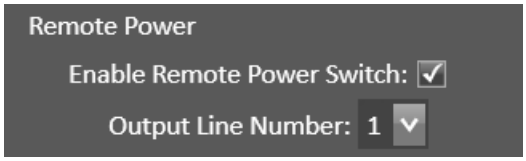
Zero Lift Axis Program Coord: This will zero the lift axis program coordinates after each Touch Off.

Rapid Move to Start Height: After the first Touch Off, this will allow the lift axis to move at a rapid rate until the Start Height when it will move at the general sensing feedrate for the remainder of the Touch Off. The "Start Height" value can be specified in CNC. This can increase productivity while cutting, particularly if using a large Safe Height.

Enable Suppression Radius: After the first Touch Off, the system will not Touch Off for subsequent cuts if starting within this radius from the prior Touch Off. The "Suppression Radius" value can be specified in CNC. This can increase productivity while cutting.

Remote Power

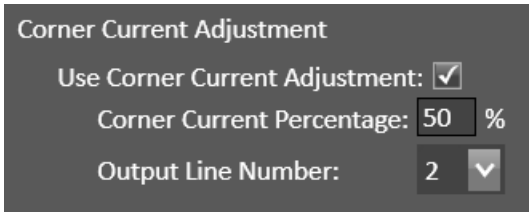
Some plasma models have an option for power to be switched on remotely.

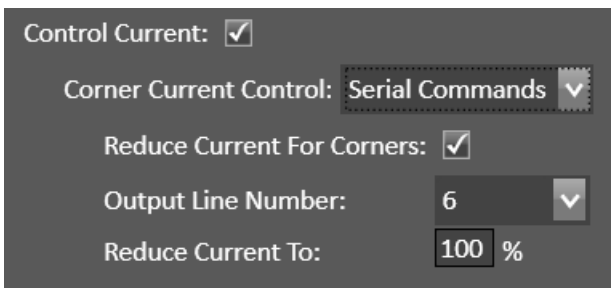


Output Line Number: Specify the output line from the controller to be used.

Corner Current

Some plasma torch models have controls to adjust the current at corners. Depending on the plasma torch model, various controls options are available.





Output Line Number: Specify the output line from the controller to be used.

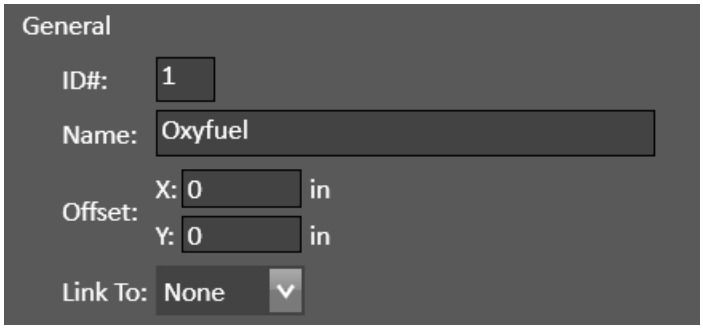
Corner Current Percentage: Specify the percentage to reduce the current.

Corner Current Control: Some plasma torch models support the use of Serial Commands or Variable Output to control corner current.

Oxyfuel

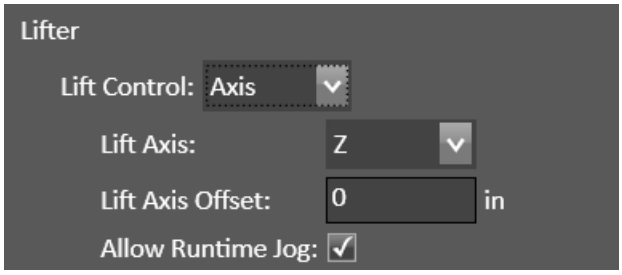
The available parameters will vary depending on the features available on your system.

General



Link To: This allows for another configured Oxyfuel to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary Oxyfuel fabrication head will be disabled, and only unique parameters will be available.

Lifter



Lifter Control:

Axis: Where you can specify the Lift Axis, it's Lift Axis Offset from other fabrication heads, and optionally Allow Runtime Jog so you can manually move the lift axis while cutting.

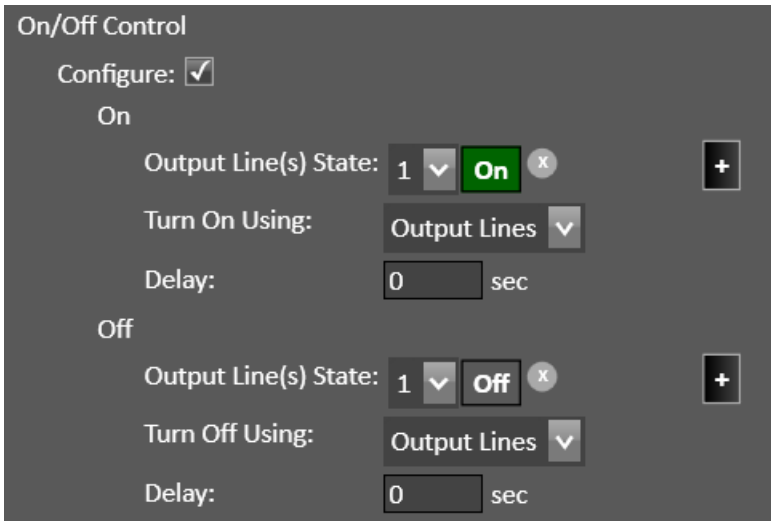
Two Position: Where you specify how to control the Up and Down positions.

Momentary: Where you specify how to control Raise On/Off and Lower On/Off.

None

On/Off Control

It is required to specify the basic on/off controls for your oxyfuel system.



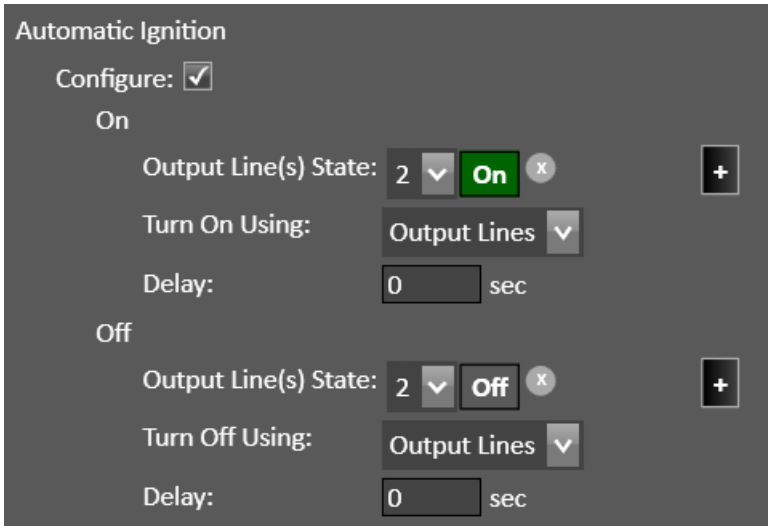
On/Off Output Line(s) State: For most systems, this will be Output Line #1 On for On and Output Line #1 Off for Off. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most oxyfuel systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Automatic Ignition

If using automatic ignition, it is required to specify the on/off controls.



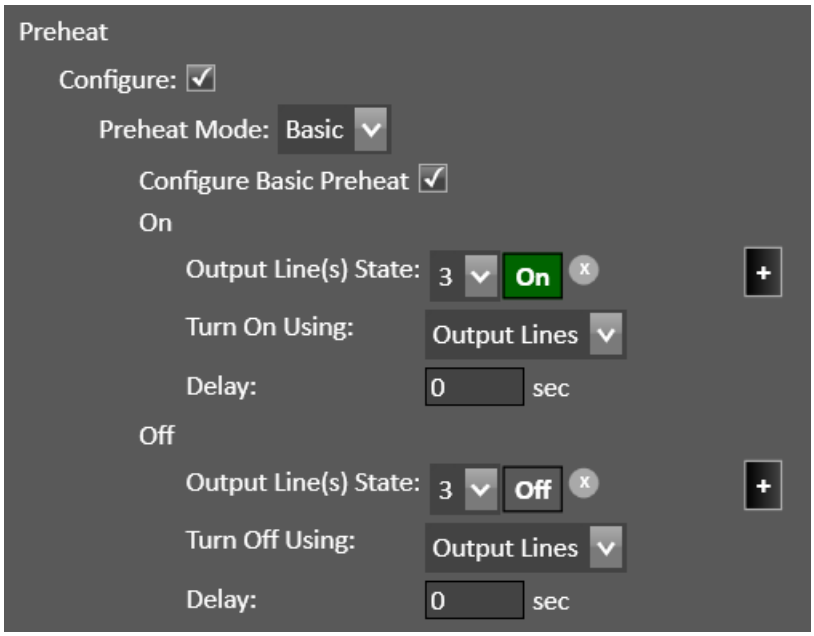
On/Off Output Line(s) State: Specify the output line(s) and state to control automatic ignition. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most oxyfuel systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Preheat

If using preheat it is required to specify the on/off controls.



Preheat Mode: If using Basic Preheat, a single set of On/Off controls will be available to configure. If using Bi-Level Preheat, there will be separate controls for Low Preheat and High Preheat.

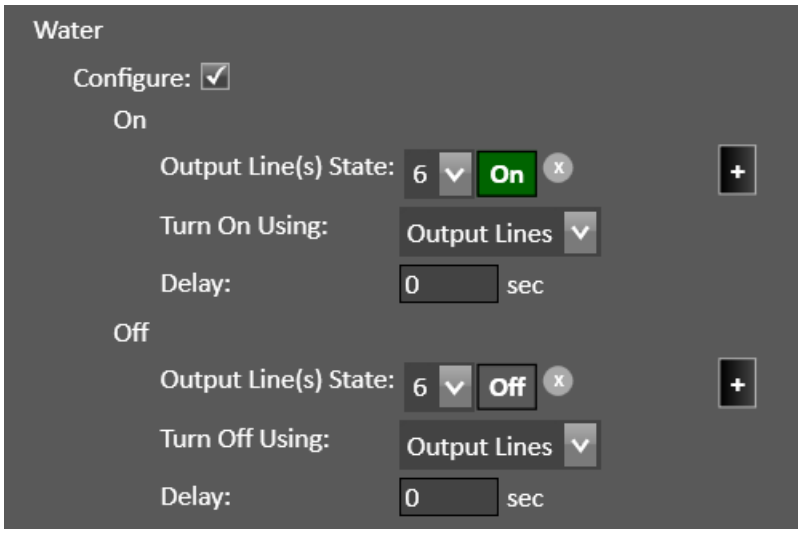
On/Off Output Line(s) State: Specify the output line(s) and state to control preheat. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most oxyfuel systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Water

When using water, it is required to specify the on/off controls.



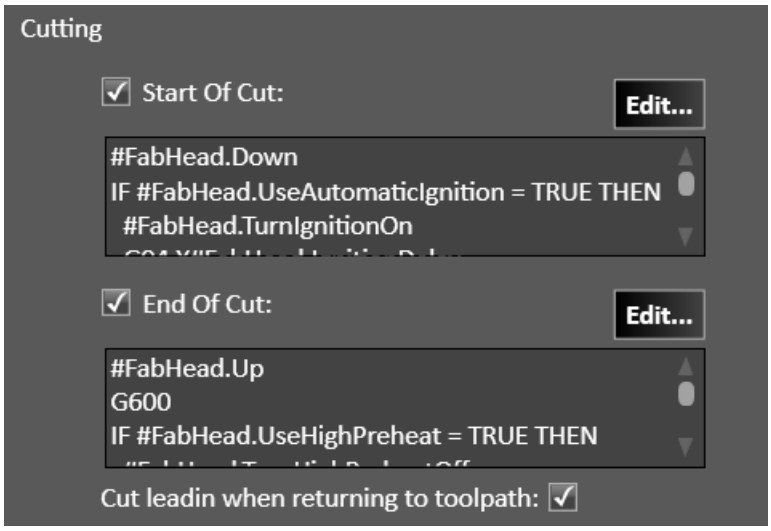
On/Off Output Line(s) State: Specify the output line(s) and state to control water. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most oxyfuel systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Cutting

Specify the commands to be executed at the Start and End of each cut in a G-Code program.



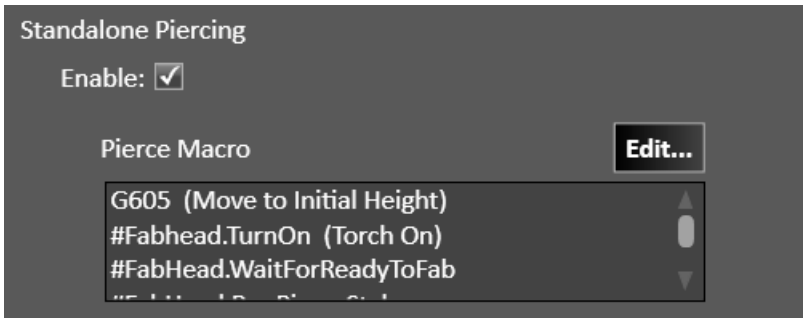
Start of Cut: For some oxyfuel systems, the default set of commands in the macro will work well. These commands will be executed automatically when starting a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: For some oxyfuel systems, the default set of commands in the macro will work well. These commands will be executed automatically when ending a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Cut leadin when returning to toolpath: Enable this feature if you want to cut a lead-in from the oxyfuel torch's current position to the toolpath when resuming a G-Code program.

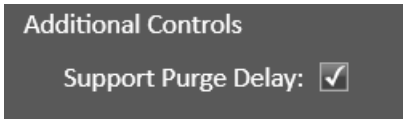
Standalone Piercing

Specify the commands to be executed for a standalone (separate) pierce in a G-Code program.



Pierce Macro: For some oxyfuel systems, the default set of commands in the macro will work well. These commands will be executed when a pierce command is in the G-Code program. Typically, this includes the same commands found in the Start of Cut macro and End of Cut macro. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Additional Controls

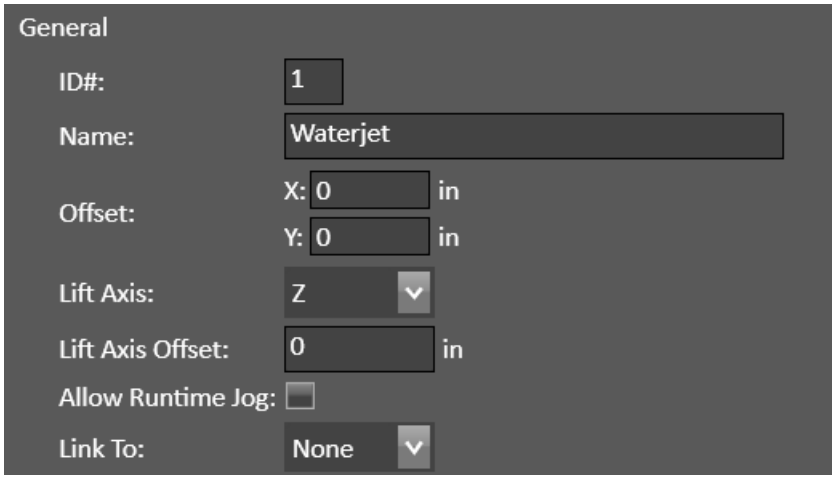


Support Purge Delay: This control option will be made available in CNC.

Waterjet

The available parameters will vary depending on the features available on your system.

General



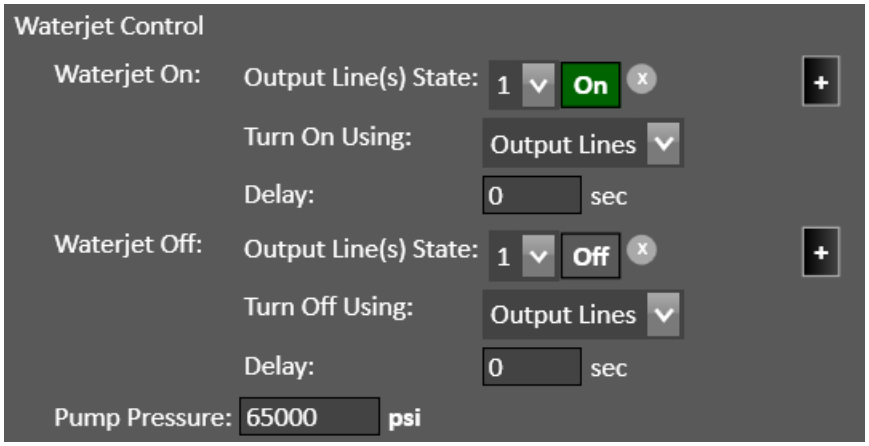
Lift Axis: This defaults to the Z axis. A different axis can be selected (W V U). A typical scenario is when multiple fabrication heads are configured, where you may also need to measure the **Lift Axis Offset** value. If your plasma system does not use a lift axis, select None. Several lift-axis-specific features will no longer be available when there is no lift axis.

Allow Runtime Jog: When enabled you can manually move the lift axis while cutting.

Link To: This allows for another configured Waterjet to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary fabrication head will be disabled, and only unique parameters will be available.

Waterjet Control

It is required to specify the basic on/off controls for your waterjet system.



Waterjet On/Off Output Line(s) State: Specify the output line(s) and state to control waterjet. Use the "+" button to add more output lines or the "x" button to delete an output line.

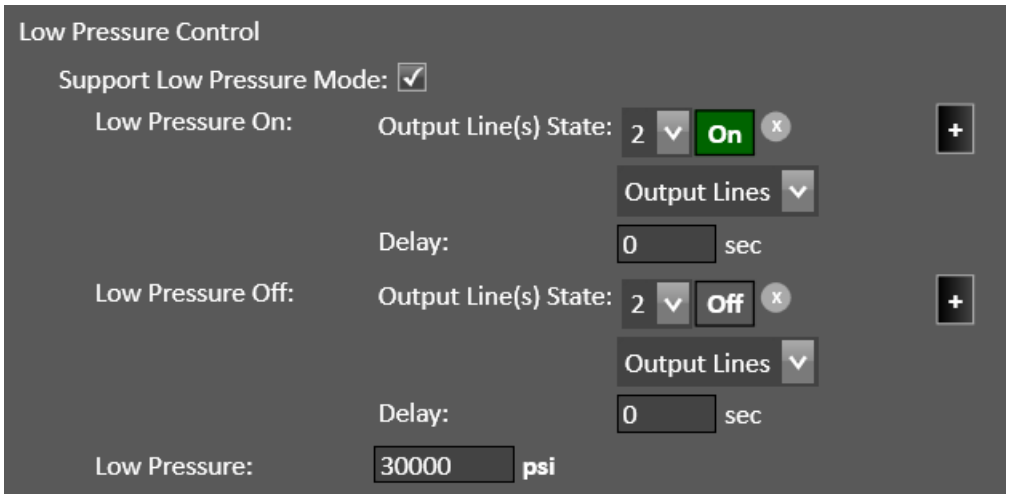
Turn On/Off Using: For most waterjet systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most waterjet systems, no delay is required.

Pump Pressure: Specify the pump pressure for your waterjet system.

Low Pressure Control

If using a low-pressure mode, it is required to specify the on/off controls.



Low Pressure On/Off Output Line(s) State: Specify the output line(s) and state to control low pressure mode. Use the "+" button to add more output lines or the "x" button to delete an output line.

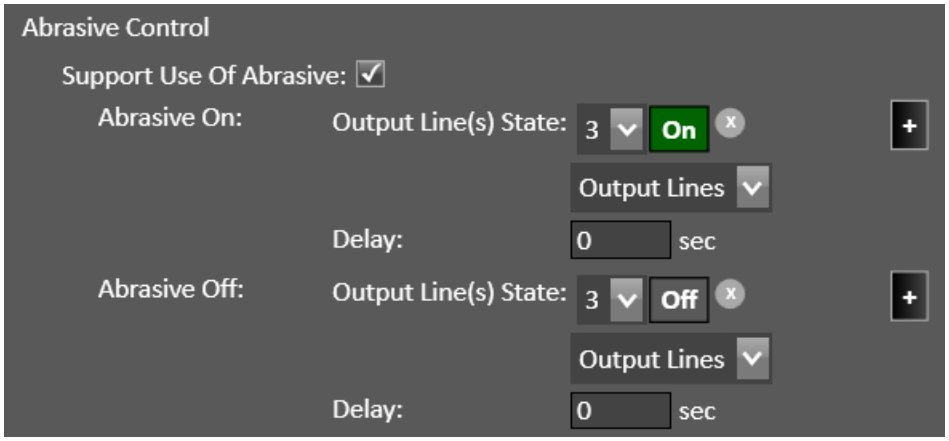
Turn On/Off Using: For most waterjet systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most waterjet systems, no delay is required.

Low Pressure: Specify the pump pressure in low pressure mode for your waterjet system.

Abrasive Control

If using a abrasive, it is required to specify the on/off controls.



Abrasive On/Off Output Line(s) State: Specify the output line(s) and state to control abrasive. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most waterjet systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most waterjet systems, no delay is required. Note: There are separate Abrasive On/Off Delay parameters that are specified in CAM and CNC.

Cutting

Specify the commands to be executed at the Start and End of each cut in a G-Code program.

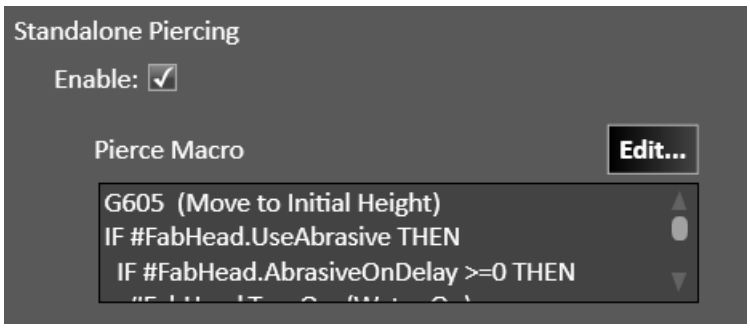
Start of Cut: For some waterjet systems, the default set of commands in the macro will work well. These commands will be executed automatically when starting a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: For some waterjet systems, the default set of commands in the macro will work well. These commands will be executed automatically when ending a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Cut leadin when returning to toolpath: Enable this feature if you want to cut a lead-in from the waterjet's current position to the toolpath when resuming a G-Code program.

Standalone Piercing

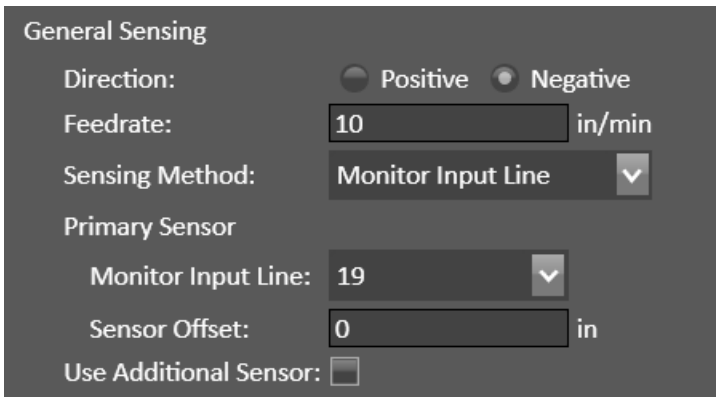
Specify the commands to be executed for a standalone (separate) pierce in a G-Code program.



Pierce Macro: For some waterjet systems, the default set of commands in the macro will work well. These commands will be executed when a pierce command is in the G-Code program. Typically, this includes the same commands found in the Start of Cut macro and End of Cut macro. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

General Sensing

Specify the sensing parameters to be used for Touch Off and Program Zero Sensing of the lift axis.



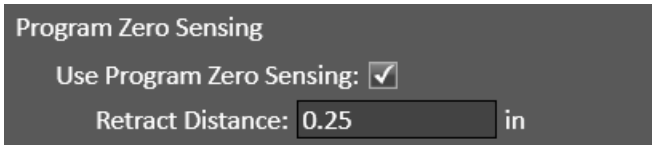
Direction: This is typically Negative, down toward the material.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 10 in/min. Setting this value too high can result in inaccurate results or other errors.

Sensing Method: This can be "Monitor Motor Current" if the lift axis uses a servo drive. You specify the **Current Limit** and a **Current Debounce**. Or this can be "Monitor Input Line" and you can use a Primary Sensor and optionally **Use Additional Sensor**. You specify the **Monitor Input Line** and **Sensor Offset**.

Program Zero Sensing

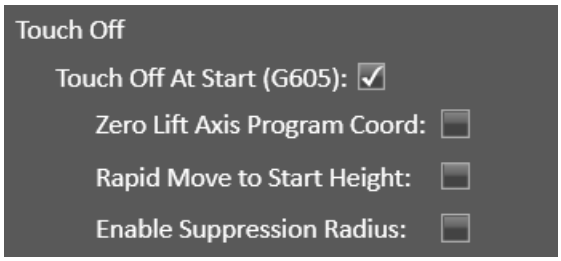
This will enable program zero sensing in the Program coordinate's Set menu.



Retract Distance: This is the distance the lift axis will retract after sensing program zero.

Touch Off

Enable the use of Touch Off (the G605 command). Controls will be displayed in CNC. If your system has advanced touch off features, those can be enabled here.



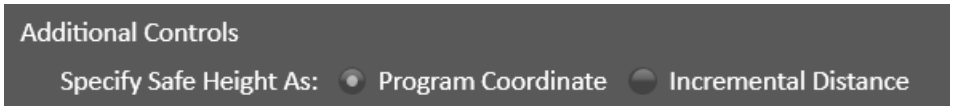
Zero Lift Axis Program Coord: This will zero the lift axis program coordinates after each Touch Off.

Rapid Move to Start Height: After the first Touch Off, this will allow the lift axis to move at a rapid rate until the Start Height when it will move at the general sensing feedrate for the remainder of the Touch Off. The "Start Height" value can be specified in CNC. This can increase productivity while cutting, particularly if using a large Safe Height.

Enable Suppression Radius: After the first Touch Off, the system will not Touch Off for subsequent cuts if starting within this radius from the prior Touch Off. The "Suppression Radius" value can be specified in CNC. This can increase productivity while cutting.

Additional Controls

The only additional controls are how to specify the Safe Height coordinates.



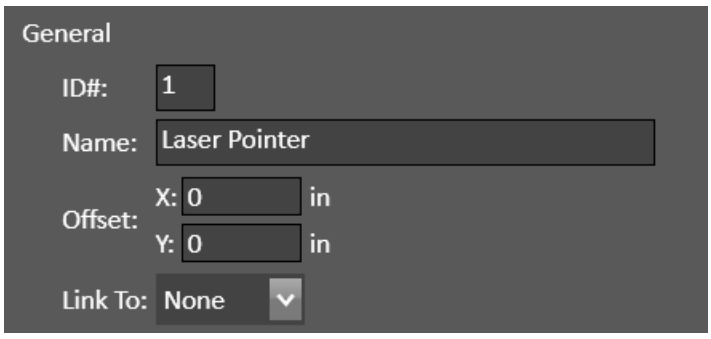
Specify Safe Height As: This is typically set to "Program Coordinate". For some systems "Incremental Distance" may be preferred.

Laser

Specify the laser configuration here.

Laser Pointer

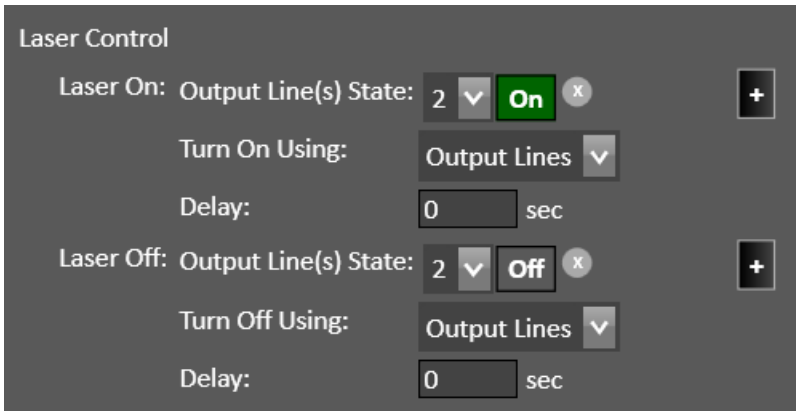
General



Link To: This allows for another configured Laser Pointer to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary fabrication head will be disabled, and only unique parameters will be available.

Laser Control

It is required to specify the basic on/off controls for your laser pointer.



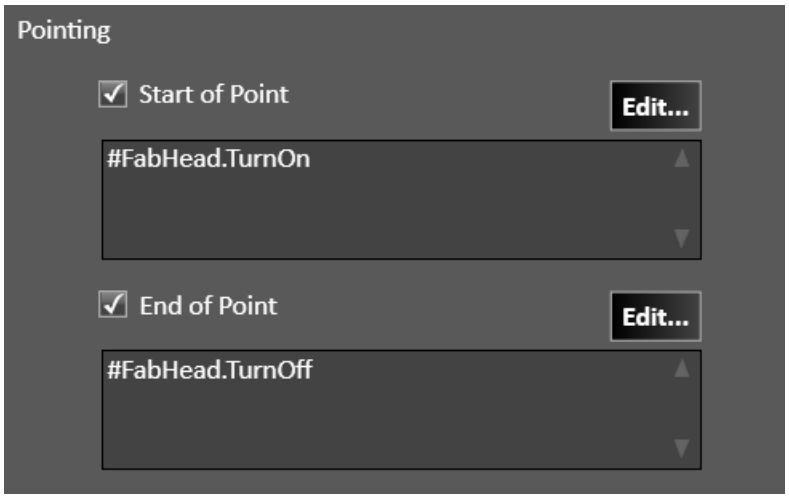
Laser On/Off Output Line(s) State: Specify the output line(s) and state to control the laser pointer. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most waterjet systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Pointing

Specify the commands to be executed at the Start and End of each cut in a G-Code program. A typical configuration is to have the laser pointer turn on and off, similar to how a fabrication head would turn on and off while cutting, which can be useful to visualize fabrication when running a G-Code program with the Laser Pointer "Locked".



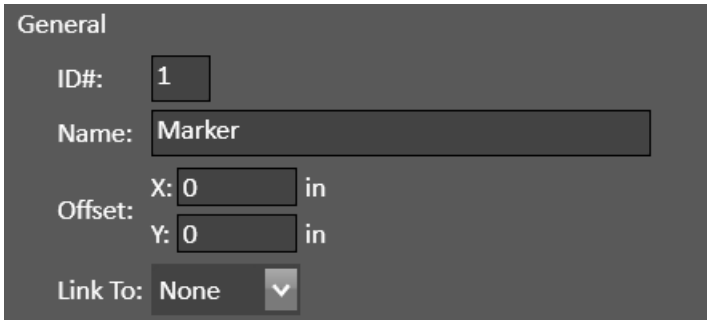
Start of Point: The default macro has commands to turn the laser pointer on. These commands will be executed automatically when starting a cut/mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Point: The default macro has commands to turn the laser pointer off. These commands will be executed automatically when ending a cut/mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Marker

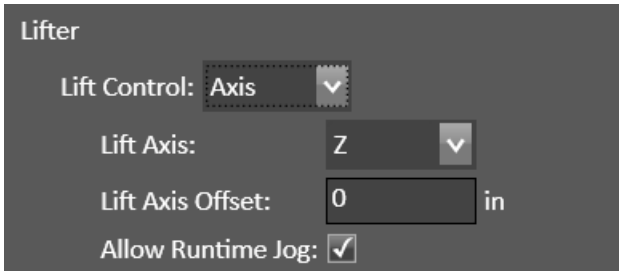
The available parameters will vary depending on the features available on your marking/scribe system.

General



Link To: This allows for another configured Marker to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary fabrication head will be disabled, and only unique parameters will be available.

Lifter



Lifter Control:

Axis: Where you can specify the Lift Axis, it's Lift Axis Offset from other fabrication heads, and optionally Allow Runtime Jog so you can manually move the lift axis while cutting.

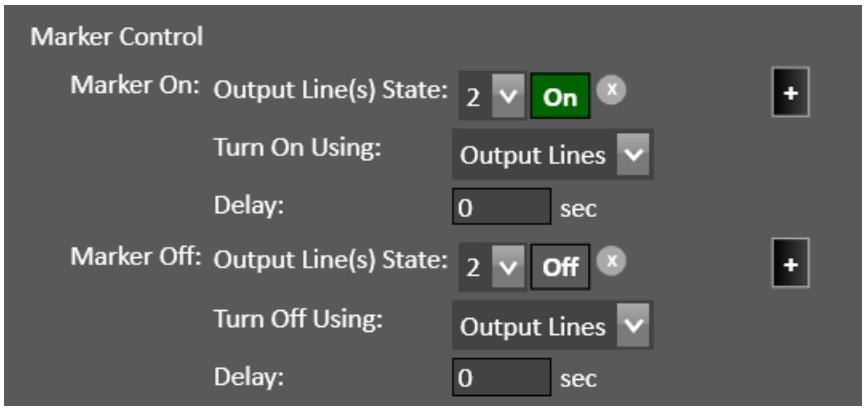
Two Position: Where you specify how to control the Up and Down positions.

Momentary: Where you specify how to control Raise On/Off and Lower On/Off.

None

Marker Control

It is required to specify the basic on/off controls for your marker system.



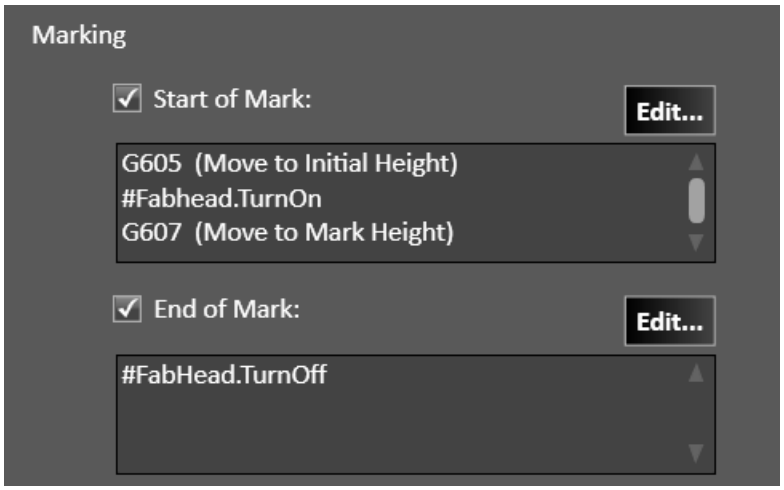
Marker On/Off Output Line(s) State: Specify the output line(s) and state to control the marker/scribe. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most marker systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Marking

Specify the commands to be executed at the Start and End of each mark in a G-Code program.

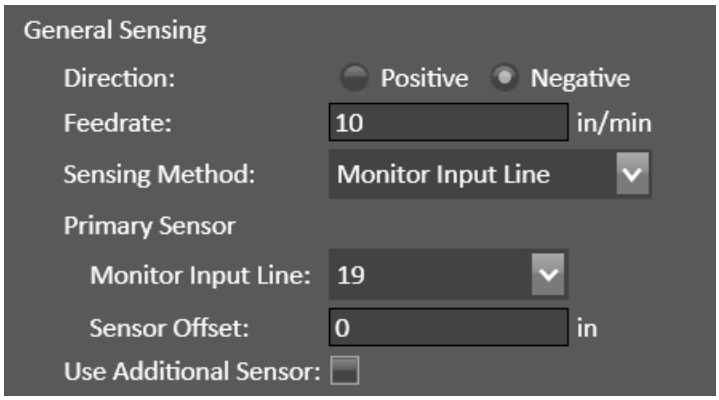


Start of Mark: For marker systems with a lift axis, the default set of commands in the macro should work well. These commands will be executed automatically when starting a mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: For most marker systems, the default set of commands in the macro will work well. These commands will be executed automatically when ending a mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

General Sensing

Specify the sensing parameters to be used for Touch Off and Program Zero Sensing of the lift axis.



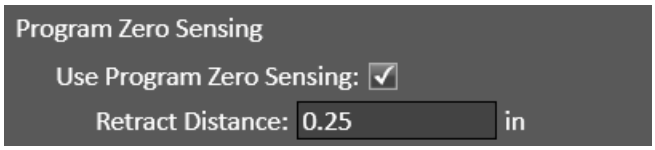
Direction: This is typically Negative, down toward the material.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 10 in/min. Setting this value too high can result in inaccurate results or other errors.

Sensing Method: This can be "Monitor Motor Current" if the lift axis uses a servo drive. You specify the **Current Limit** and a **Current Debounce**. Or this can be "Monitor Input Line" and you can use a Primary Sensor and optionally **Use Additional Sensor**. You specify the **Monitor Input Line** and **Sensor Offset**.

Program Zero Sensing

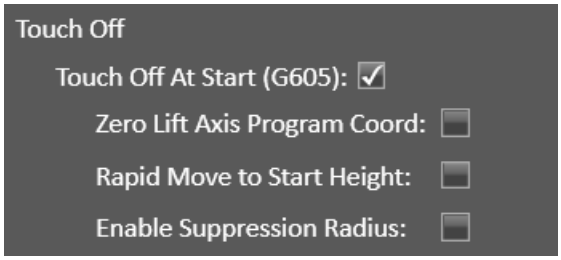
This will enable program zero sensing in the Program coordinate's Set menu.



Retract Distance: This is the distance the lift axis will retract after sensing program zero.

Touch Off

Enable the use of Touch Off (the G605 command). Controls will be displayed in CNC. If your system has advanced touch off features, those can be enabled here.



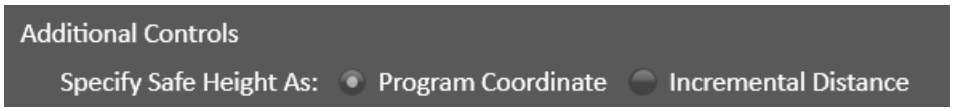
Zero Lift Axis Program Coord: This will zero the lift axis program coordinates after each Touch Off.

Rapid Move to Start Height: After the first Touch Off, this will allow the lift axis to move at a rapid rate until the Start Height when it will move at the general sensing feedrate for the remainder of the Touch Off. The "Start Height" value can be specified in CNC. This can increase productivity while cutting, particularly if using a large Safe Height.

Enable Suppression Radius: After the first Touch Off, the system will not Touch Off for subsequent cuts if starting within this radius from the prior Touch Off. The "Suppression Radius" value can be specified in CNC. This can increase productivity while cutting.

Additional Controls

The only additional controls are how to specify the Safe Height coordinates.

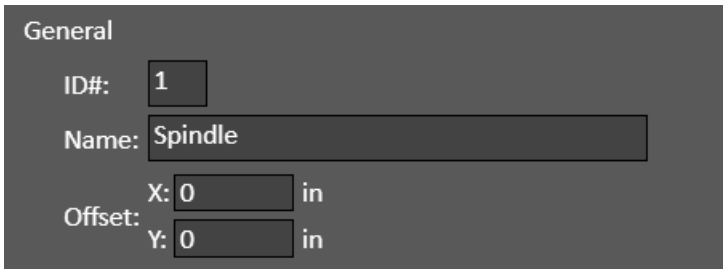


Specify Safe Height As: This is typically set to "Program Coordinate". For some systems "Incremental Distance" may be preferred.

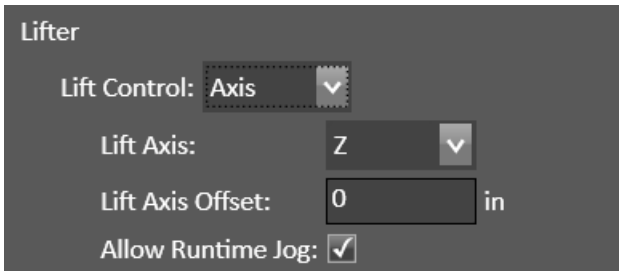
Mill

The available parameters will vary depending on the features available on your mill/spindle/router system.

General



Lifter



Lifter Control:

Axis: Where you can specify the Lift Axis, it's Lift Axis Offset from other fabrication heads, and optionally Allow Runtime Jog so you can manually move the lift axis while cutting.

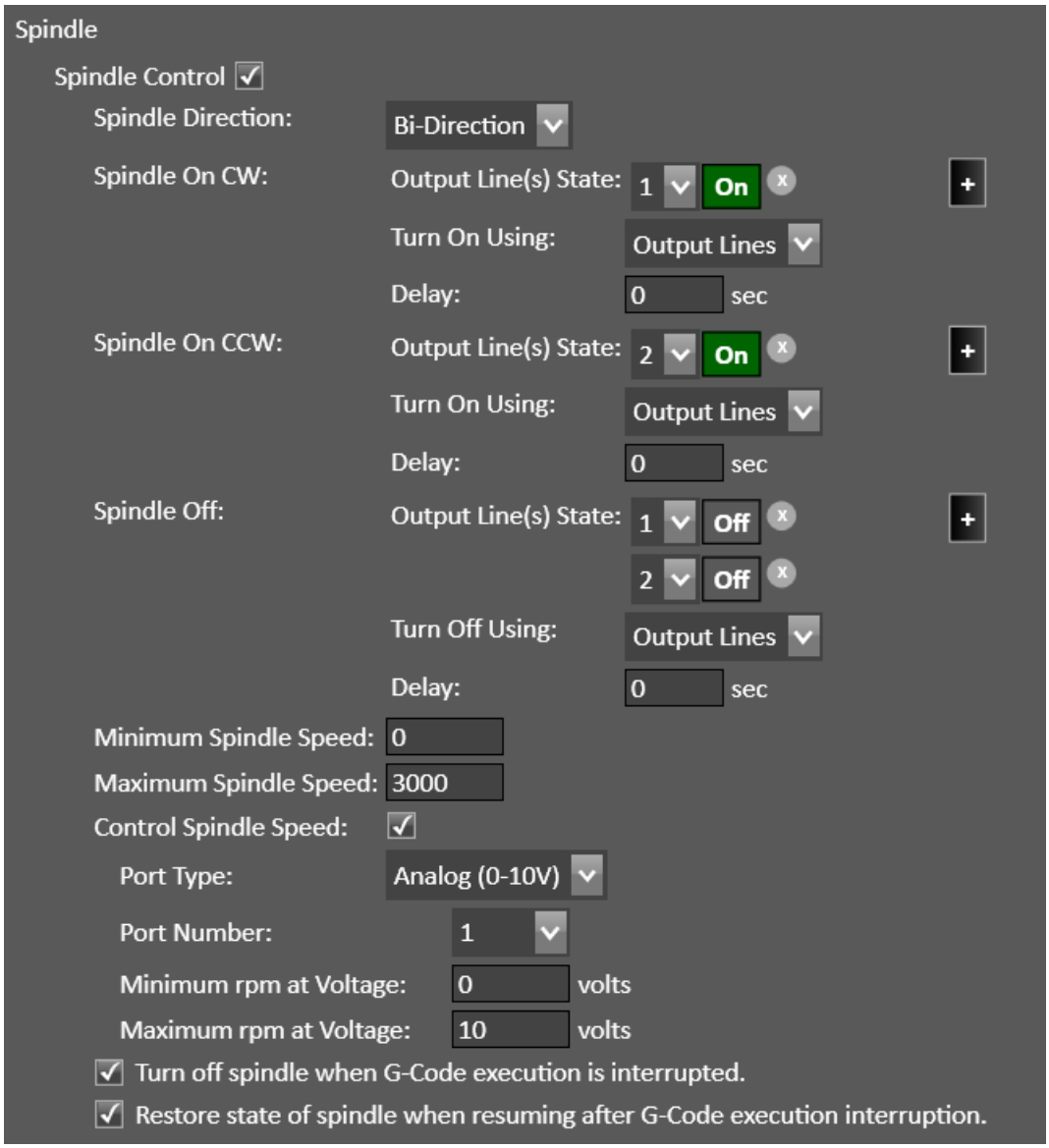
Two Position: Where you specify how to control the Up and Down positions.

Momentary: Where you specify how to control Raise On/Off and Lower On/Off.

None

Spindle Control

If a spindle is to be controlled there are several options available. It can be a simple on/off control, a directional clockwise/counter-clockwise control, and the spindle speed could be controlled via a variable output.



Spindle Direction: Choose "Single Direction" for a simple on/off control. Choose "Bi-Direction" for clockwise and counter-clockwise controls.

Spindle On/Off Output Line(s) State: Specify the output line(s) and state to control the spindle. Use the "+" button to add more output lines or the "x" button to delete an output line. Note: When using bi-directional spindle control, specify the output lines to turn off both the clockwise and counter-clockwise states.

Turn On/Off Using: For most spindle systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: Specify a delay duration if desired.

Minimum/Maximum Spindle Speed: These spindle speed values are important when using a variable spindle speed controller (configured below).

Control Spindle Speed: Enable when using a variable spindle speed controller.

Port Type: This indicates the type of signal for controlling the spindle speed. Typical systems will use "Analog" where a voltage varies from 0V to 10V. You will also need to specify the **Port Number** that is being used: 1 or 2.

Other, less typical, options include "PWM" where the output is at a specified frequency and the duty cycle varies, or "PFM" where the output is at a specified duty cycle and the frequency varies.

NOTE: After making a change to the Port Type, an application restart may be required.

Minimum/Maximum rpm at ____: The spindle speeds will correspond to a minimum and maximum variable output, controlled proportionally to the desired spindle speed per the minimum and maximum spindle speed values defined (above).

Turn off spindle when G-Code execution is interrupted. Enable if you want the spindle to be automatically turned off when the execution of a G-Code program is interrupted.

Restore state of spindle when resuming after G-Code execution interruption. Enable if you want the spindle state to be restored upon resuming a G-Code program after the G-Code program execution had been interrupted.

Cutting

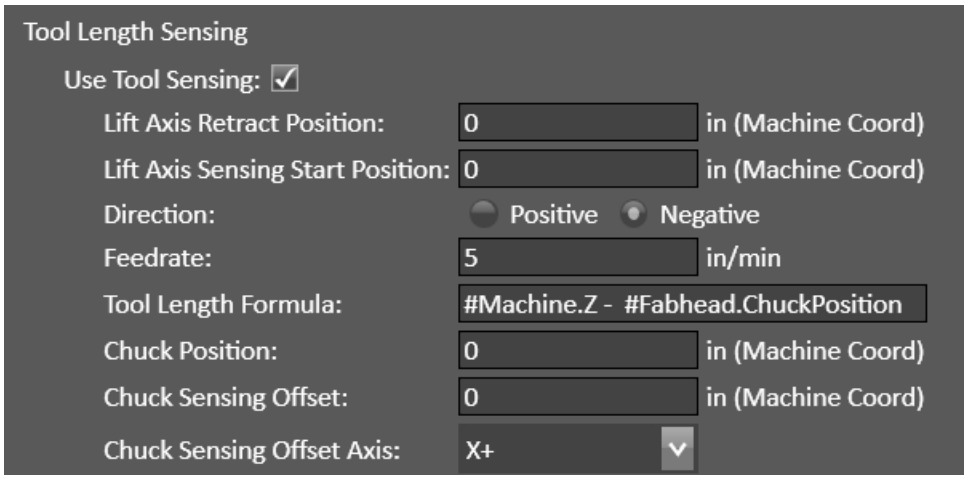
Specify the commands to be executed at the Start and End of each cut in a G-Code program.

Start of Cut: These commands will be executed automatically when starting a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: These commands will be executed automatically when ending a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Tool Length Sensing

Specify the parameters for tool length sensing.



Lift Axis Retract Position: The lift axis will rapid move to this position (typically up away from the sensor) before moving X and Y to the sensor position.

Lift Axis Sensing Start Position: The lift axis will rapid move to this position (typically down toward the sensor) before sensing.

Direction: This is typically Negative, down toward the sensor.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 5 in/min. Setting this value too high can result in inaccurate results or other errors.

Tool Length Formula: For the currently loaded Tool, its Tool Offset value will be updated based on the calculated result of this formula. The formula is typically: #Machine.Z - #FabHead.ChuckPosition. For more information about the available commands refer to the Programming Reference document.

Chuck Position: This position value can be used for the Tool Length Formula calculation of the Tool Offset.

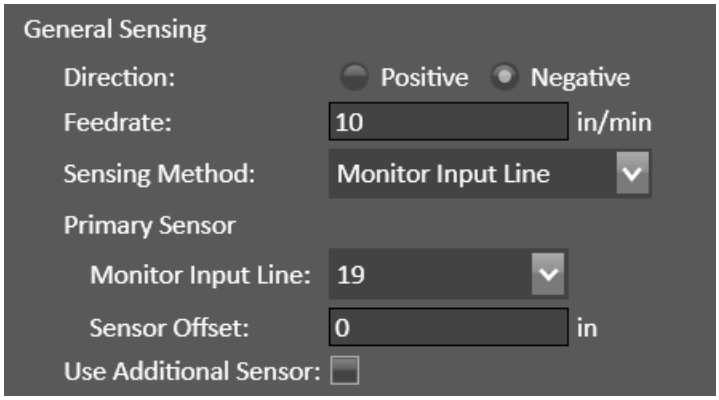
Chuck Sensing Offset: When sensing the chuck, an X or Y offset may need to be accounted for.

Chuck Sensing Offset Axis: Defines the axis and direction of the Chuck Sensing Offset.

General Sensing

Specify the sensing parameters to be used for Touch Off and Program Zero Sensing of the lift axis.

Note: These are different parameters, not used for Tool Length Sensing.



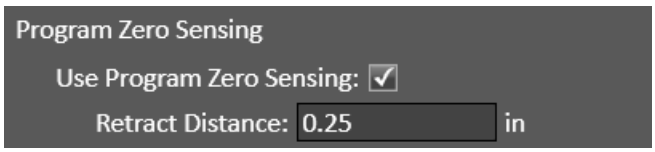
Direction: This is typically Negative, down toward the material.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 10 in/min. Setting this value too high can result in inaccurate results or other errors.

Sensing Method: This can be "Monitor Motor Current" if the lift axis uses a servo drive. You specify the **Current Limit** and a **Current Debounce**. Or this can be "Monitor Input Line" and you can use a Primary Sensor and optionally **Use Additional Sensor**. You specify the **Monitor Input Line** and **Sensor Offset**.

Program Zero Sensing

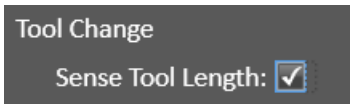
This will enable program zero sensing in the Program coordinate's Set menu.



Retract Distance: This is the distance the lift axis will retract after sensing program zero.

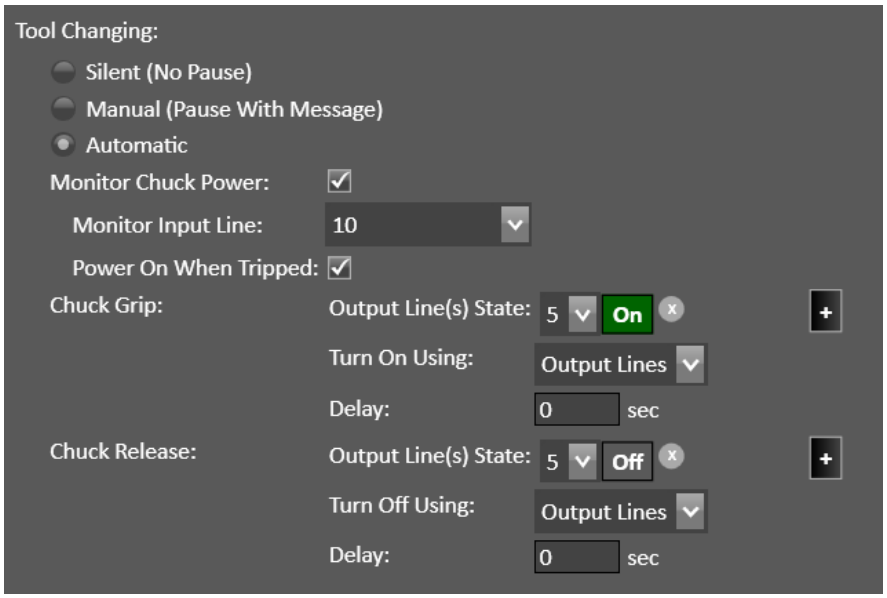
Tool Change

For Manual or Automatic Tool Changing while running a G-Code program, it will sense the length of the tool after each tool change.



Tool Changing

Tool changes can be handled in several ways. It could be a manual tool change or an automatic tool change for several types of tool rack configurations.



Silent (No Pause) The tool change occurs but the G-Code program does not stop, and no physical tool change occurs. This mode can be useful if using one tool for a project, or if running a G-Code program as a test run while not actually cutting material.

Manual (Pause with Message) The G-Code program will be paused and a message to the operator will be displayed to change the tools.

Automatic A tool change macro (M06) will be executed, as defined in the Tool Changer section of this document. Some of the features used in the tool change macro can be configured in that Tool Changer section and some of the mill-specific parameters can be configured below:

Monitor Chuck Power: Enable if you want to monitor an input line that indicates the presence of tool chuck power. Typically, this is used to monitor air pressure for the spindle chuck. If the input line indicates no power, motion halts and an error message is displayed. This prevents a crash when the system attempts to change tools.

Monitor Input Line: The input line to monitor tool chuck power.

Power On When Tripped: The state of the input line when the tool chuck has power.

Chuck Grip/Release Output Line(s) State: Specify the output line(s) and state to control the chuck. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most chucks, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Tangential Knife

General

The available parameters will vary depending on the tangential knife model and the features available on your system.

Lift Axis: This defaults to the Z axis. A different axis can be selected (W V U). A typical scenario is when multiple fabrication heads are configured, where you may also need to measure the **Lift Axis Offset** value. If your system does not use a lift axis, select None. Several lift-axis-specific features will no longer be available when there is no lift axis.

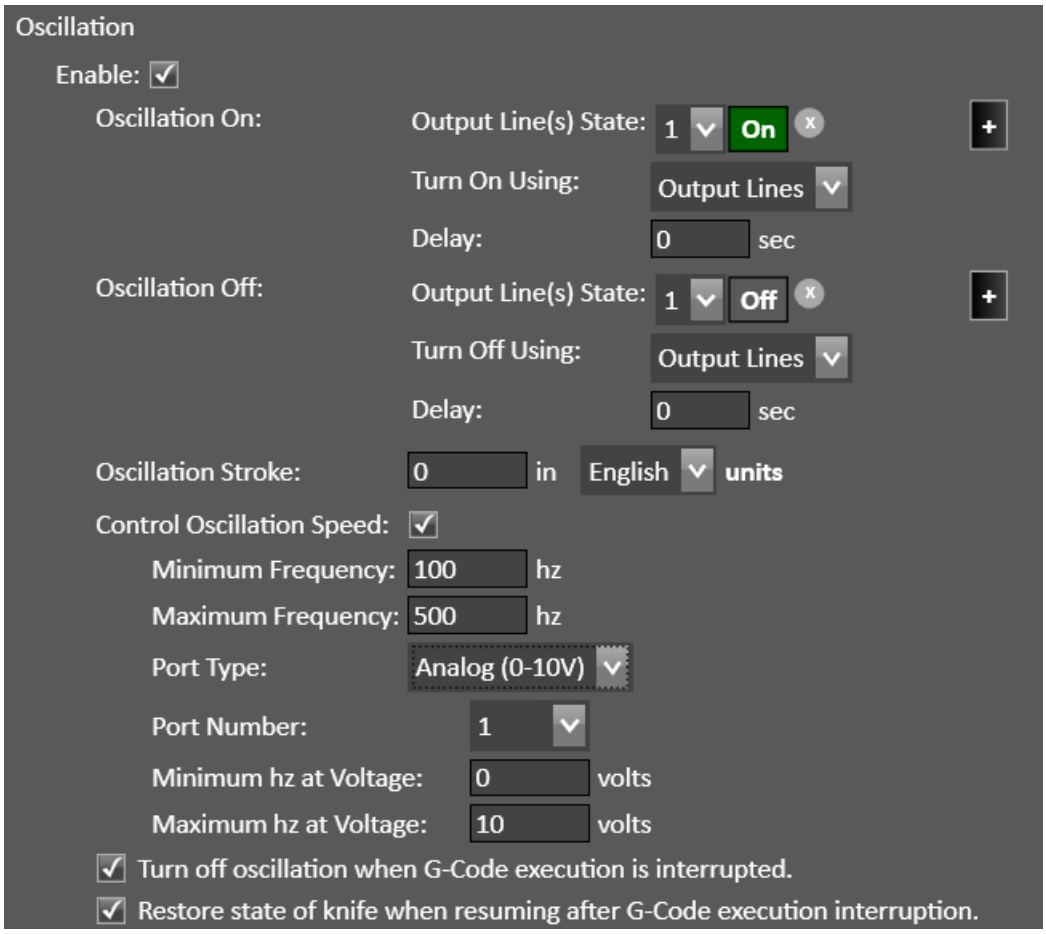
Rotary Axis: This defaults to the C axis. A different rotary axis can be selected (A B C).

Knife 0 Direction: This defines the cutting direction of the knife's rotary zero position relative to the X and Y coordinates of the machine.

Rotary Axis Offset to 0: Specify any offset of the rotary axis' zero position relative to the knife's zero position.

Oscillation

If using an oscillating knife, the on/off controls need to be specified.



Oscillation On/Off Output Line(s) State: Specify the output line(s) and state to control the knife. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most spindle systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: Specify a delay duration if desired.

Oscillation Stroke: Specify the oscillation stroke length.

Control Oscillation Speed: Enable when using a variable output controller.

Minimum/Maximum Frequency: These oscillation frequency values will correspond with the speed controller (configured below).

Port Type: This indicates the type of signal for controlling the oscillation frequency. Typical systems will use "Analog" where a voltage varies from 0V to 10V. You will also need to specify the **Port Number** that is being used: 1 or 2.

Other options include "PWM" where an output is at a specified frequency and the duty cycle varies from 0% to 100%, or "PFM" where an output is at a specified duty cycle and the frequency varies.

Minimum/Maximum hz at ____: The oscillation frequency will correspond to a minimum and maximum variable output, controlled proportionally to the desired oscillation frequency per the minimum and maximum frequency values defined (above).

Turn off oscillation when G-Code execution is interrupted. Enable if you want the knife oscillation to be automatically turned off when the execution of a G-Code program is interrupted.

Restore state of knife when resuming after G-Code execution interruption. Enable if you want the knife oscillation to be restored upon resuming a G-Code program after the G-Code program execution had been interrupted.

Cutting

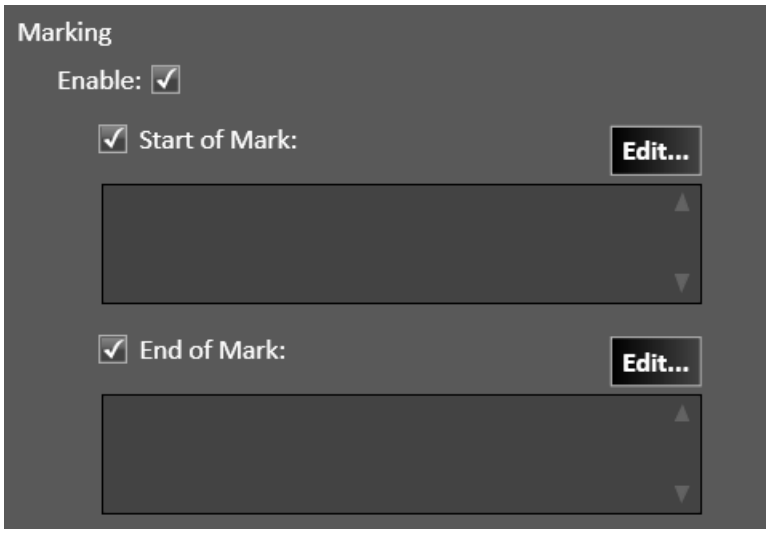
Specify the commands to be executed at the Start and End of each cut in a G-Code program.

Start of Cut: These commands will be executed automatically when starting a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Cut: These commands will be executed automatically when ending a cut in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Marking

Specify the commands to be executed at the Start and End of each mark in a G-Code program.

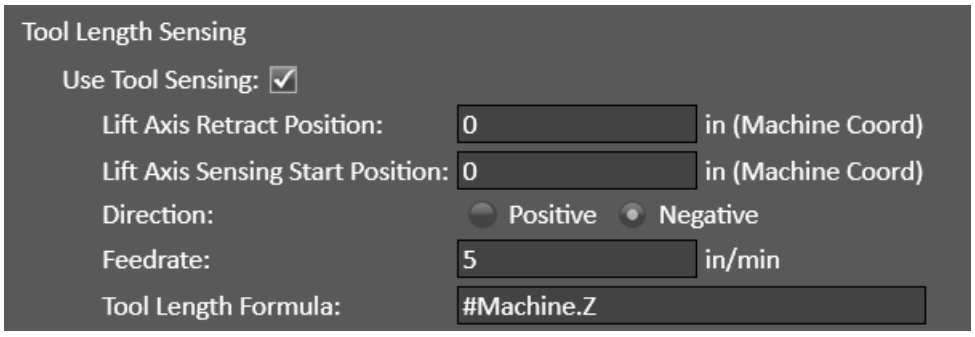


Start of Mark: These commands will be executed automatically when starting a mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

End of Mark: These commands will be executed automatically when ending a mark in the G-Code program. The macro can be edited for unique systems. For more information about the available commands refer to the Programming Reference document.

Tool Length Sensing

Specify the parameters for tool length sensing.



Lift Axis Retract Position: The lift axis will rapid move to this position (typically up away from the sensor) before moving X and Y to the sensor position.

Lift Axis Sensing Start Position: The lift axis will rapid move to this position (typically down toward the sensor) before sensing.

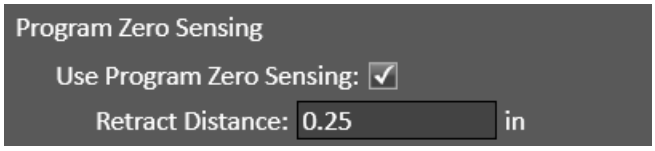
Direction: This is typically Negative, down toward the sensor.

Feedrate: This is the rate the lift axis moves while sensing. A typical value is 5 in/min. Setting this value too high can result in inaccurate results or other errors.

Tool Length Formula: For the currently loaded Tool, its Tool Offset value will be updated based on the calculated result of this formula. The formula is typically: #Machine.Z. For more information about the available commands refer to the Programming Reference document.

Program Zero Sensing

This will enable program zero sensing in the Program coordinate's Set menu.



Retract Distance: This is the distance the lift axis will retract after sensing program zero.

Tool Changing

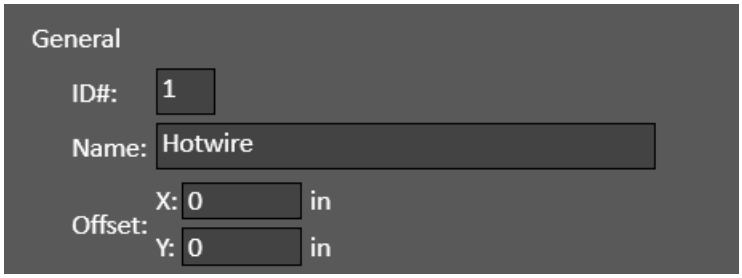
Silent (No Pause) The tool change occurs but the G-Code program does not stop, and no physical tool change occurs. This mode can be useful if using one tool for a project, or if running a G-Code program as a test run while not actually cutting material.

Manual (Pause with Message) The G-Code program will be paused and a message to the operator will be displayed to change the tools.

Hotwire

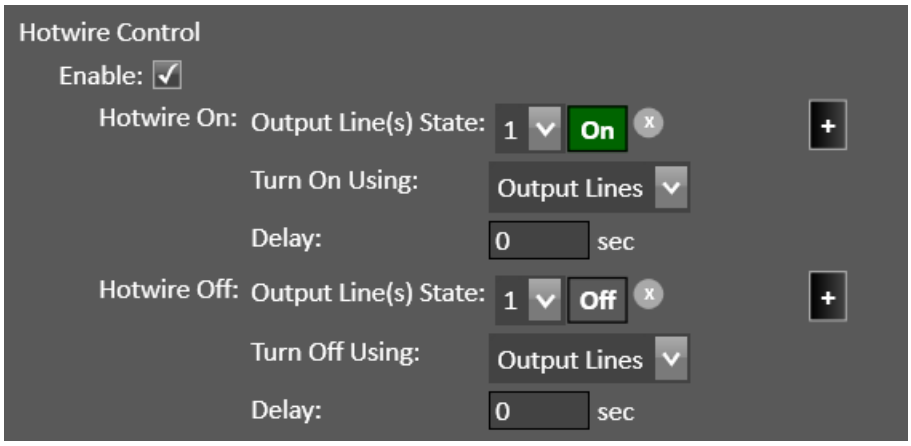
Note: A Hotwire fabrication head can be configured for use in CNC however CAD/CAM does not yet support hotwire g-code generation.

General



Hotwire Control

It is required to specify the basic on/off controls for your hotwire.



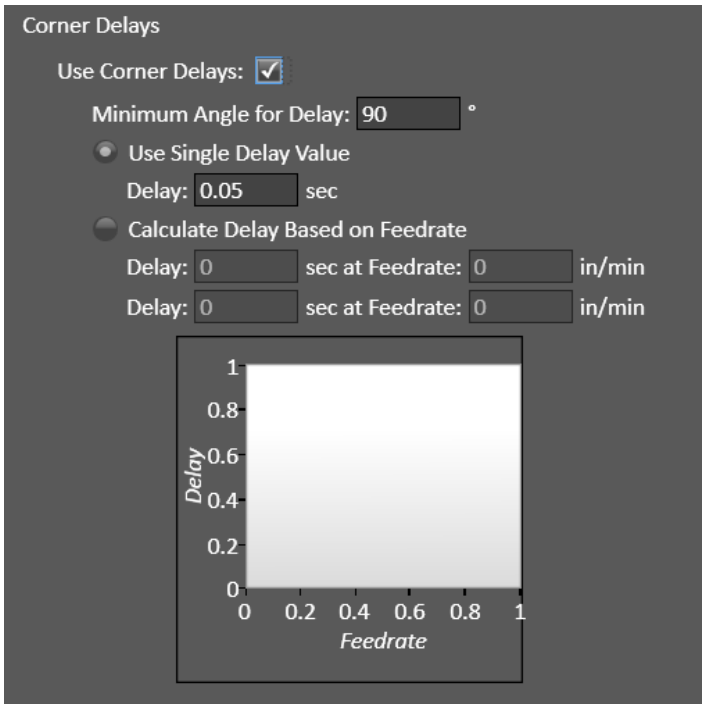
Hotwire On/Off Output Line(s) State: Specify the output line(s) and state to control the hotwire. Use the "+" button to add more output lines or the "x" button to delete an output line.

Turn On/Off Using: For most systems, this will be "Output Lines". For advanced/custom systems, you can use a "Custom Macro". The custom macro should include at least one command that sets the Output Line(s) State appropriately.

Delay: For most systems, no delay is required.

Corner Delays

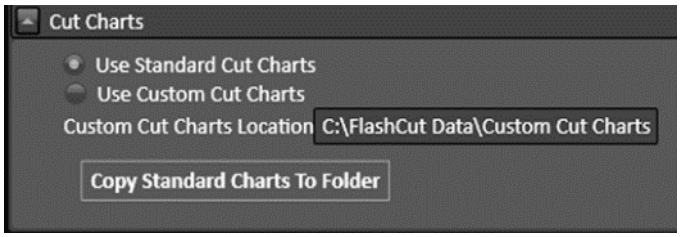
For some hotwire systems, corner delays are beneficial.



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.



Use Standard Cut Charts

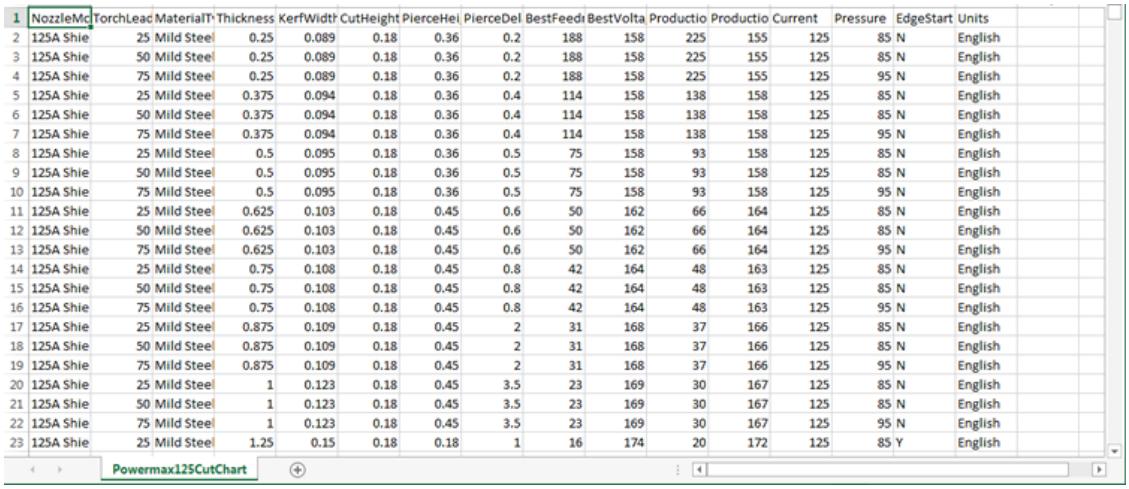
Use the pre-loaded cut charts with recommended settings such as feedrate, pierce height, cut height and THC set point.

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.

An example is shown below:



	NozzleMc	TorchLead	MaterialT	Thickness	KerfWidtF	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

Fabrication Head Change

Pause on Fabrication Head Change

Select to enable. This feature is useful if the lifter needs to be swapped to mount a different fabrication head.

Axes

Motor Line/Axis Mapping

FlashCut provides three 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 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.

In general, axes X Y Z are always linear axes. Axes W V U could be linear or rotary depending on the specific machine. If they are rotary axes, the W V U axes are called A B C.

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 homing switch, the total distance each sub axis retracts from its homing switch after making contact is:

home switch offset for the axis + sub axis offset.

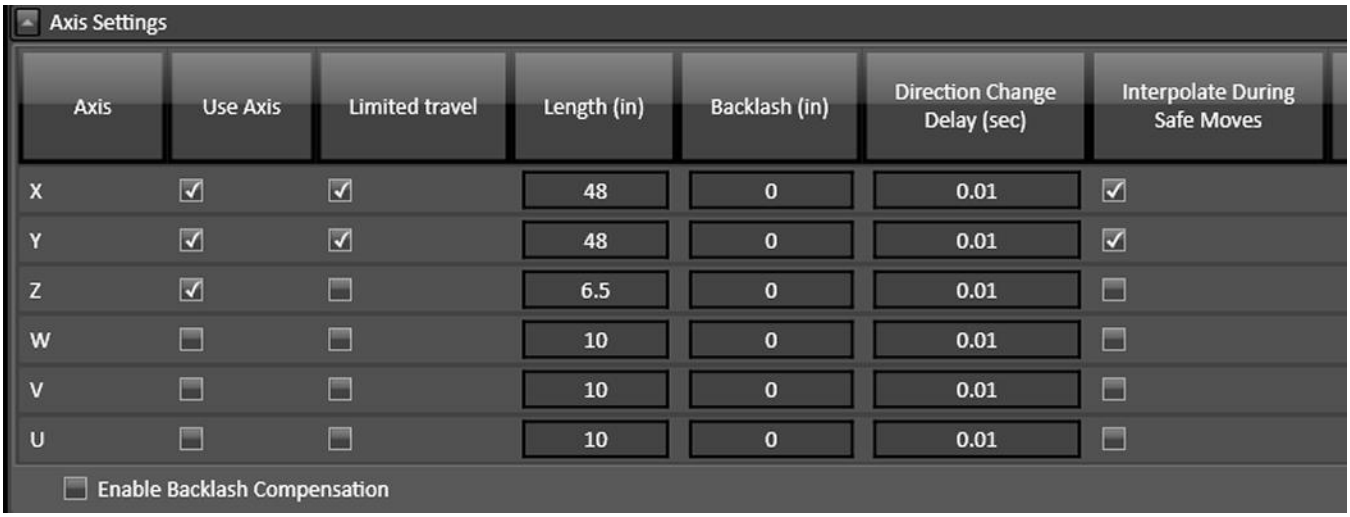
If one homing switch is used for multiple sub axes, or if multiple sub axes are not being used, this sub axis offset value should be zero.

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 counter-clockwise.

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.

Axis Settings



Enables you to define whether or not an axis is being used and if there are physical limits to each axis in use.

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.

Use Axis

Indicates whether or not the axis is used on your machine tool.

Limited Travel

Indicates whether or not the axis travel is limited. Many rotary axes do not have limited travel.

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.

Backlash

Enter the backlash value for this axis here. See below for a more detailed explanation of backlash.

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.

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 configuration fields properly, you can make the sequence safe for your machine tool configuration.

Enable Backlash Compensation

When checked, FlashCut will use backlash compensation for all direction changes.

On Backlash Compensation

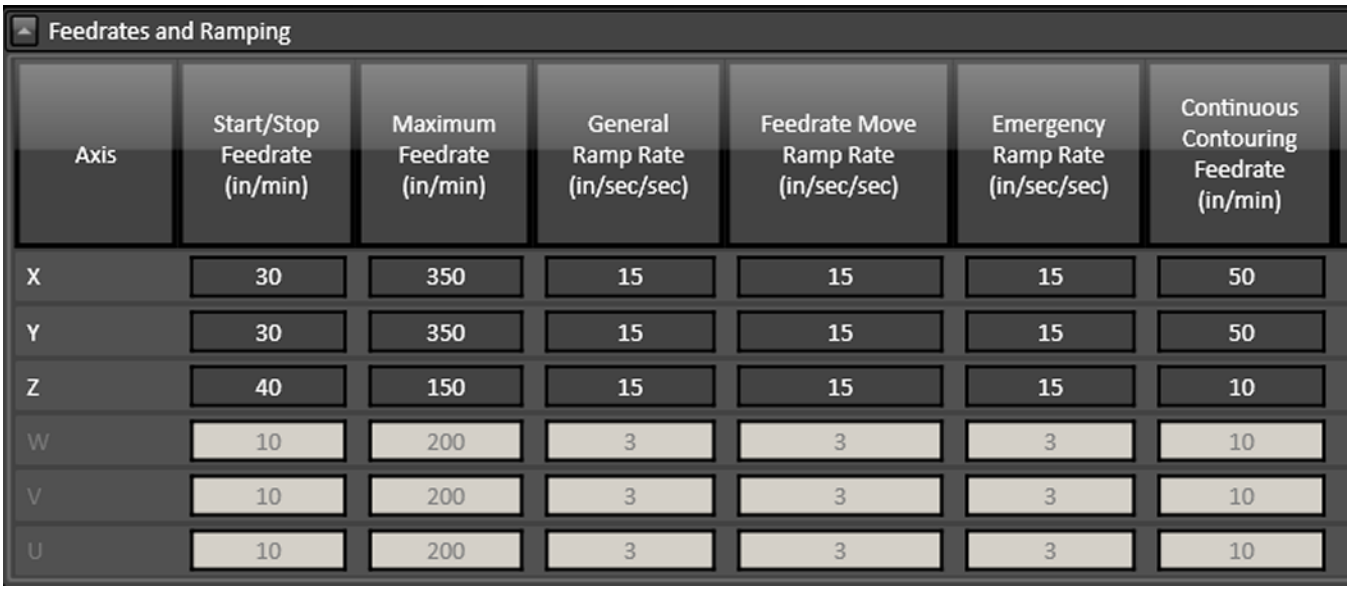
Backlash Compensation: When an axis changes direction, a small turn of the motor to move the drive mechanism through any mechanical play, before resuming motion.

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.

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.

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:

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.

Start/Stop Feedrate (in/min)

Sets the initial and ending feedrate for ramped moves.

To set the start/stop feedrates:

a. 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.)

b. All settings in these steps assume inch units; if your machine is set for metric units, you should multiply these settings by 25.

c. 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.

d. 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.

e. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.

f. 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.

g. Choose Feedrate/Ramping from the Configuration menu.

h. Enter 70% of the value you found in the Start/Stop Feedrate text box for the X axis, and then choose OK.

i. Repeat this process for all axes.

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.

a. Enter 1 in/sec2 in the General Ramping Rate text box for the X axis. (This is a conservative ramping rate.)

b. Set the Maximum Feedrate to 500 in/min (or the highest value your settings allow), and then choose OK.

c. 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.

d. 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.

e. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.

f. 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.

g. Choose Feedrate/Ramping from the Configuration menu.

h. Enter 70% of the highest no-slip feedrate you found in the X axis Maximum Feedrate text box.

i. Repeat this process for all axes.

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.

a. Enter 1 in/sec2 in the General Ramping Rate text box for the X axis. (This is a conservative ramping rate.)

b. Choose OK.

c. 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.

d. 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.

e. Repeat the above steps until you determine an optimal General Ramping Rate for the X axis.

f. Choose Feedrate/Ramping from the Configuration menu.

g. Enter 70% of the highest no-slip ramping rate you found in the X axis General Ramping Rate text box.

h. Copy the General Ramping Rate to the Feedrate Move Ramping Rate text box. If you determine that your system would benefit from a lower ramping rate for feedrate (cutting) moves, enter a lower Feedrate Move Ramping Rate.

i. Repeat this process for all axes.

Note that once the optimal ramping rate is determined for each axis, you may want to re-test the Maximum Feedrates to see if they can be set any higher.

Feedrate Move Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for feedrate moves.

Emergency Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration used in emergency operations, e.g. "Feed Hold (Emergency)".

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.

Advanced Axis Definition

These are advanced settings. The values should not be modified from default for most fabrication systems.

Point Move Linear Interpolation

Determines if the motion for the axis will be included in the interpolated move for a Point move or G28/G29/G30 command.

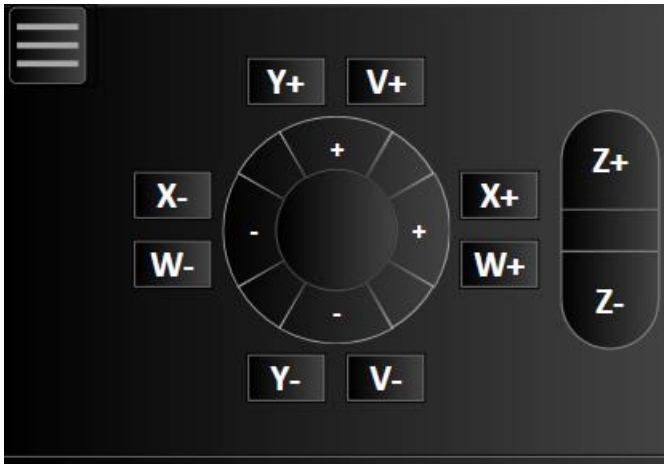
Parallel To

Indicates the orientation of the axis. For rotary axes this setting indicates the axis of rotation.

Axis Group

Some machine tools use two separate groups of axes to cut a part. FlashCut considers each pair of axes to be an axis group. When executing a move, FlashCut sets the motor speeds such that the faster-moving end of the wire (axis group) moves at the specified feedrate.

For instance, a hotwire machine uses a pair of perpendicular axes to move each end of the wire. You could have W Parallel To X as Axis Group 2 and V Parallel To Y as Axis Group 2.



Drive Parameters

Axis Drive Parameters

The axis drive parameters window displays the following information:

For each axis:

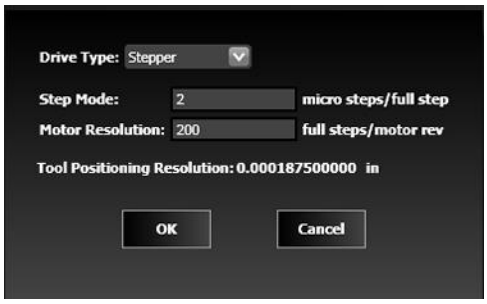
- The current Control Parameters and Drive Mechanism are displayed.
- The **Tool Positioning Resolution** is displayed. This is the length of axis movement per Signal Generator pulse, automatically calculated from the control and drive mechanism settings.

  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 drop-down menu for each axis allows you to edit the control parameters, edit the drive mechanism, or set it to be the same as another axis.

Edit Control Parameters...

Opens the Control Parameters dialog box. Select Stepper or Servo from the Driver Type pull-down menu.

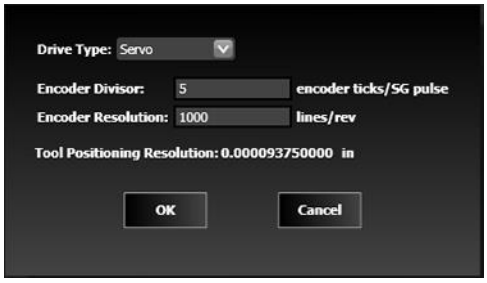
- **Stepper**



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 1.8° stepper motor has 200 full steps per revolution; a 0.9° stepper motor has 400 full steps per revolution.

- **Servo**



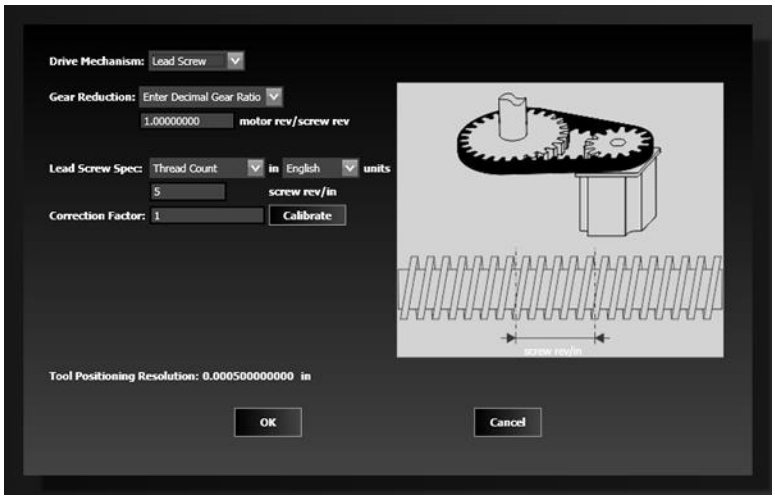
Encoder Divisor – The number of encoder pulses that correspond to one pulse from the Signal Generator.

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.

Edit Drive Mechanism...

Opens the Drive Mechanism dialog box. Select a drive mechanism from the Drive Mechanism pull-down menu.

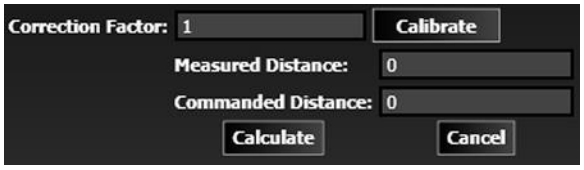
- **Lead Screw**



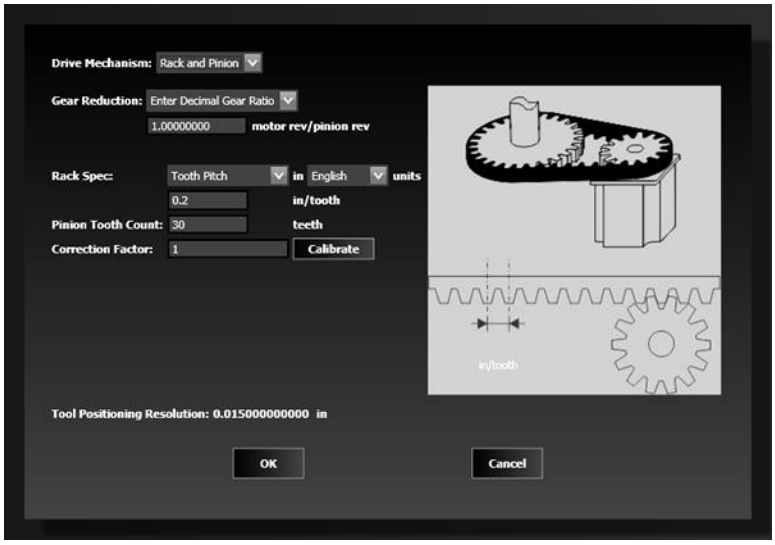
Gear Reduction – Enter the parameters for any gearing between the motor and lead screw. A decimal gear ratio of 1.0 indicates no gearing.

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**

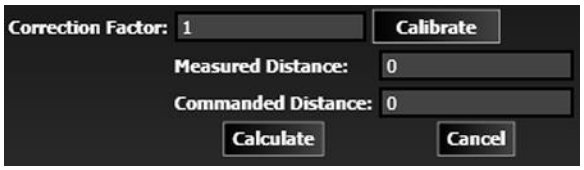


Gear Reduction – Enter the parameters for any gearing between the motor and pinion. A decimal gear ratio of 1.0 indicates no gearing.

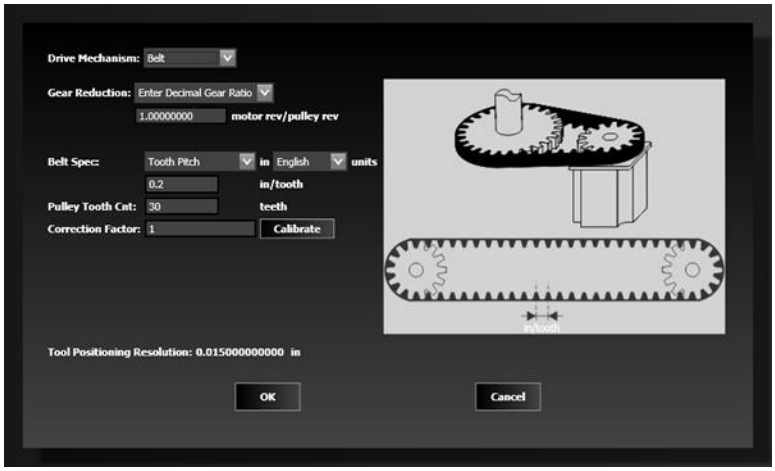
Rack Specification – Enter the parameters for the rack.

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**

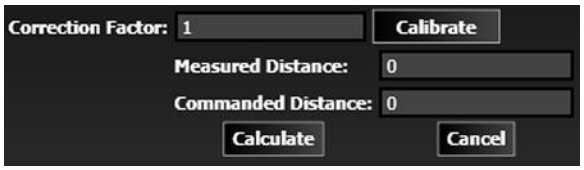


Gear Reduction – Enter the parameters for any gearing between the motor and drive pulley. A decimal gear ratio of 1.0 indicates no gearing.

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**

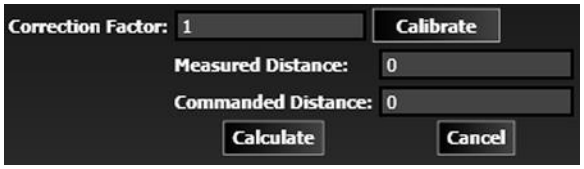
Drive Ratio – Enter the distance moved per motor revolution or the number of motor revolutions per unit distance.

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 calculates 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 ratio 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.



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.

Rotary Axis Arrangement

Click the arrow next to Rotary Axis Configuration to display a drop-down list box.

You may choose from the following configuration options:

None

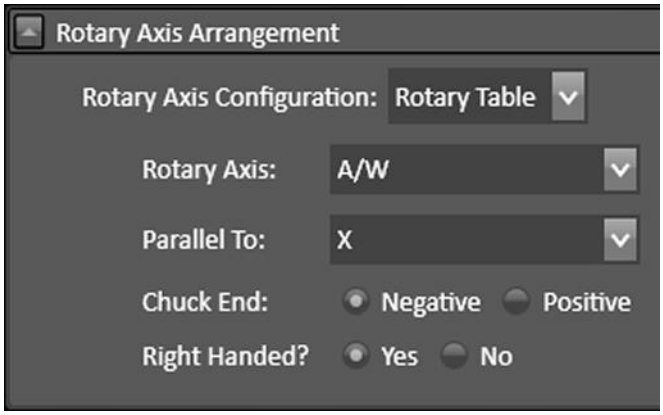
Select **None** if you are not using a rotary axis.

Other

Select **Other** if you are using some other configuration.

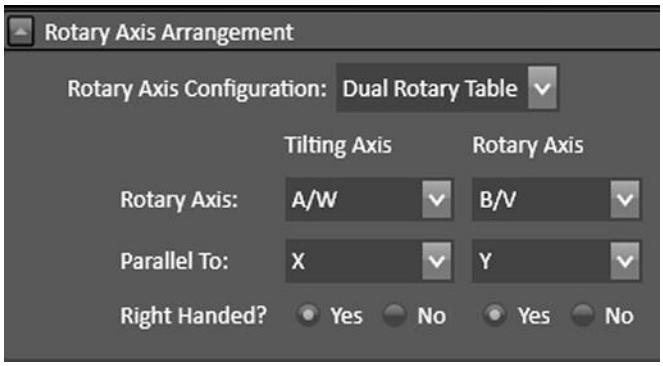
Rotary Table

Select Rotary Table if you have only one rotary axis. **Right Handed** indicates whether or not the rotary axis obeys the right hand rule. If you curl your 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 right-handed.



Dual Rotary Table

Select Dual Rotary Table if you are using two rotary axes built into a single table. You must define a Rotary Axis for each.



General

These options are part of the rotary table configuration screens.

- Use offset for rotary zero.

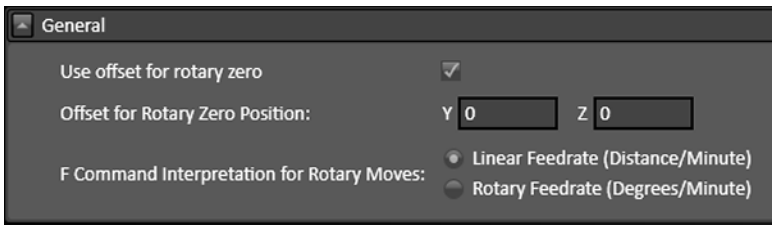
  If checked, define the offset below.

- Offset for Rotary Zero Position

  Indicates the fixture offset for the rotary zero position.

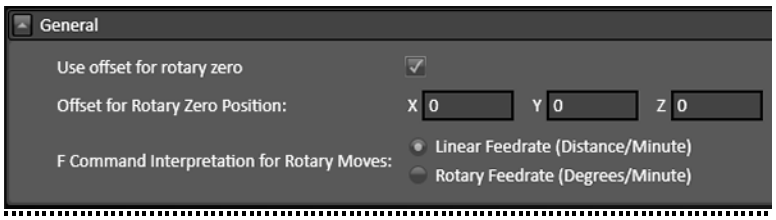
  - 4-Axis machining

    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.

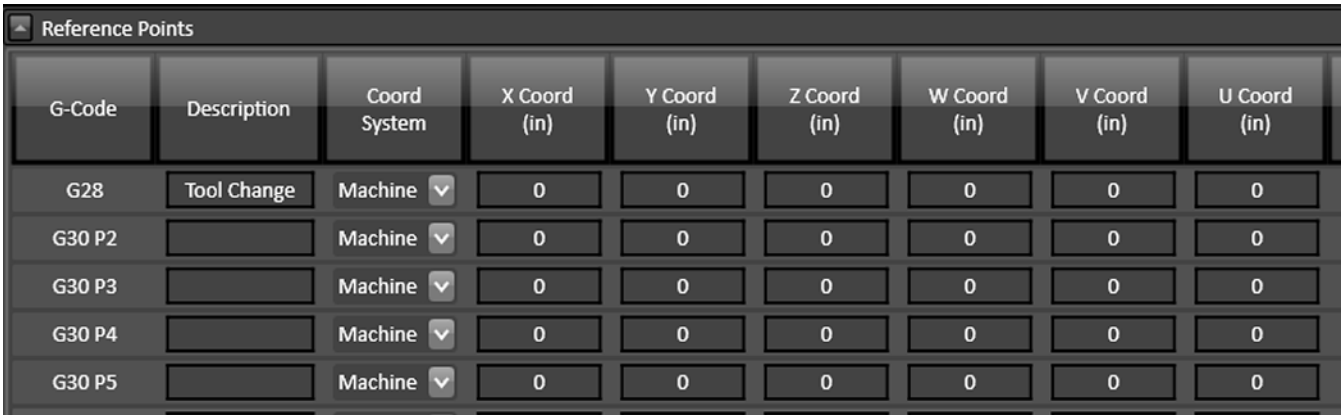
- F Command Interpretation for Rotary Moves

  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.

Reference Points

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

This field identifies the command in a G-Code file that moves the machine to the point (not editable).

Description

Enter a description for the point. For G28, the standard description is “Tool Change”.

Coord System

Choose between machine and program coordinates.

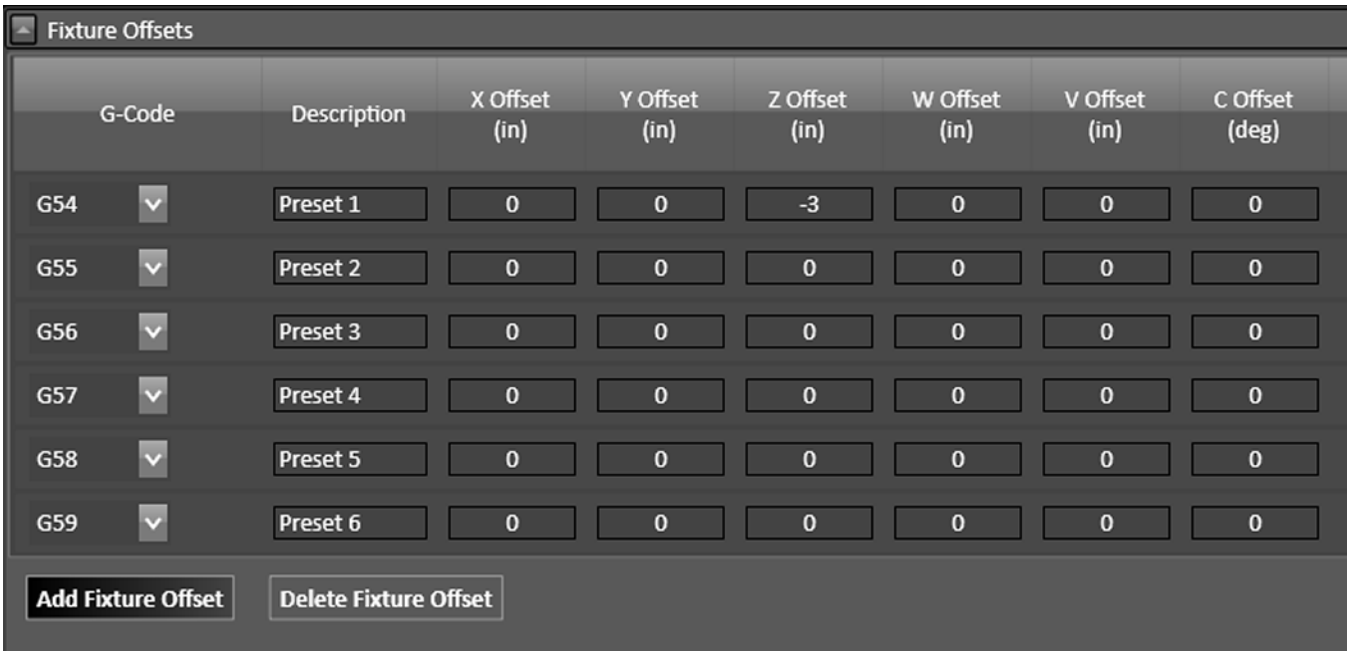
X, Y, Z, W, V, U Coordinates

Enter the coordinate for each axis.

Fixture Offsets

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.



G-Code

This field identifies the 'G' command in a G-Code file that sets Program Zero to the offset (not editable).

Description

Enter a description for the offset.

X, Y, Z, W, V, C Offset

For each axis, enter the offset from machine zero to program zero (i.e., the machine coordinate when the machine tool is at the desired program zero location).

Homing

Axis Settings

For each axis, the following settings are available:

Home Axis

Select to configure homing for the axis.

Home End

The end of the axis at Machine Zero (Home).

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.

Home Switch Offset

The distance each axis backs away from the home switch after the switch is tripped during homing.

Homing Rate

The feedrate at which the machine tool will move when locating home switches.

Homing Tolerance

The distance allowed from the original Machine Zero location to the new Machine Zero location found during homing.

Execute M-Codes Before / After

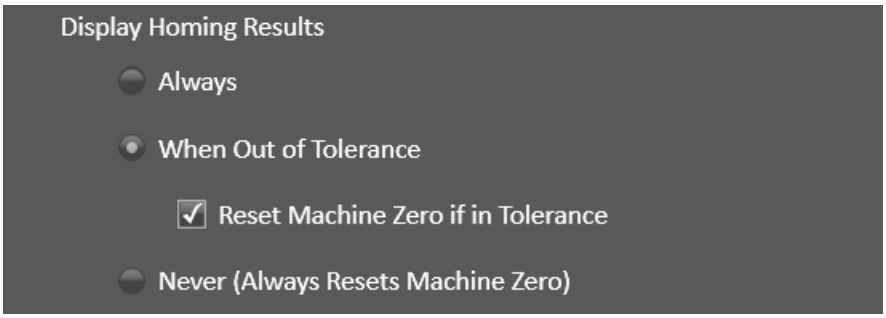
You may specify M-Codes to execute before and after each axis homes. For more information on defining M-Codes see **M-Code Definitions**.

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).

Homing Initiated by User

These settings define when to display re-homing results when the homing process is initiated by the user via the Home menu in CNC.



Always

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.

When Out of Tolerance

Tells FlashCut to display the results of re-homing, if the discrepancy from the previous Machine Zero location is outside the Homing Tolerance for any axis.

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.

Never

Tells FlashCut to not display the results of re-homing, even if the discrepancy from the previous Machine Zero location is outside the Homing Tolerance for any axis. Machine Zero will always be reset to the new location.

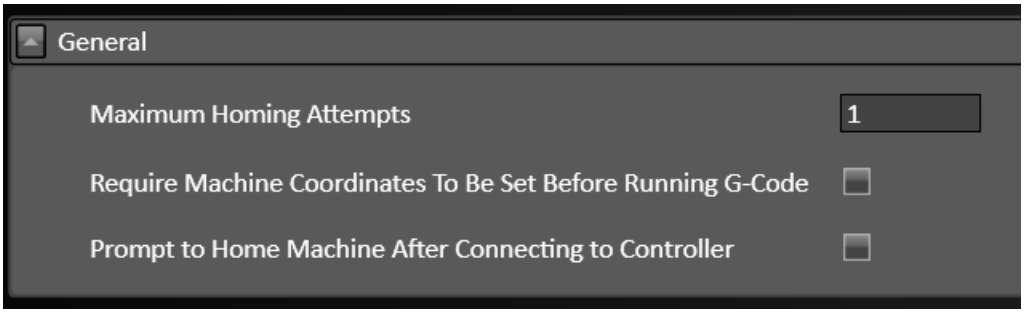
Homing Initiated While Running G-Code

These settings define when to display re-homing results when the homing process is initiated while running a G-Code program or macro (G27 command). They work the same as the user-initiated homing described above.

Automatic Macros

Commands can be executed before and after homing. These commands will only execute when homing is user-initiated via the Home menu in CNC; these commands will not execute when homing is initiated via a G27 command.

General



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.

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. A message will be displayed if not.

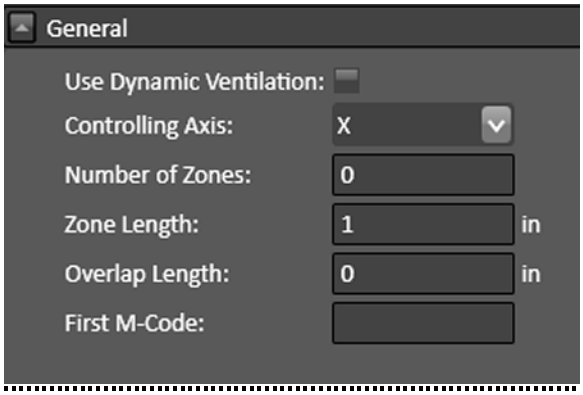
Prompt to Home Machine After Connecting to Controller

After Connecting to the controller, a message will be displayed to ask if it's safe to proceed with homing.

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.



Controlling Axis

Sets the axis along which the ventilation zones are defined.

Number of Zones

Sets the number of zones.

Zone Length

Sets the length of each zone along the controlling axis.

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.

First M-Code

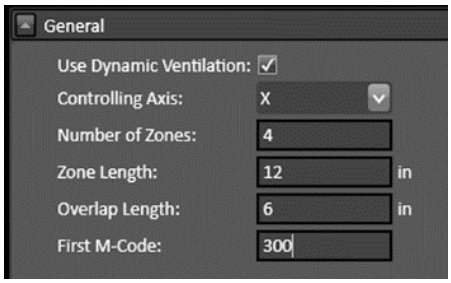
Enter the first M-Code here. The first M-Code is the code that opens the damper in zone one. Note that **codes are consecutive**: the second M-Code will close the damper for zone one, and so on.

The first M-Code cannot use the numbers 0, 1, 2, 30, 98, 99, 100, 101, 102, 103. The first M-Code must be in the list of defined M-Codes.

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 1-foot zones along the X axis, with a recommended overlap of 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.

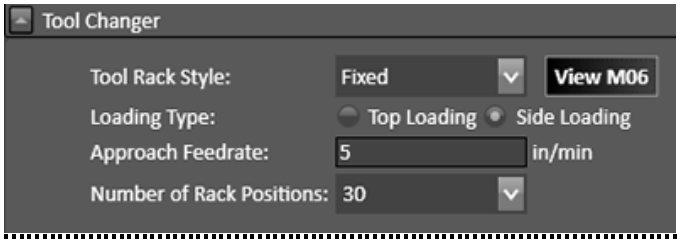
When the torch starts, the dynamic ventilation system will be active, and the damper in the first zone will be open. When the torch moves to a position 6 inches or less from the boundary of the adjacent zone, the damper in the second zone will open, and that zone will also be active. As the torch moves 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.

Tool 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 the fabrication head settings for the Mill.

Tool Changer



Tool Rack Style

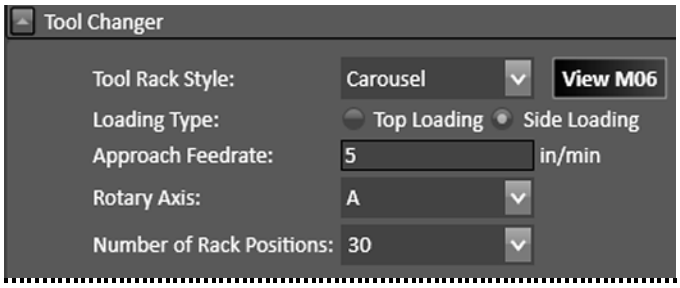
Select Fixed, Carousel, or Custom

Loading Type

Select Top or Side Loading

Approach Feedrate

Feedrate for all moves to the Lift Axis Grip and Release positions



Rotary Axis

Select axis for Carousel/Custom types.

Number of Rack Positions

Numbers refers to Tool Library entries.

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.

Tool Rack Positions

Set the following parameters for each tool rack position in machine coordinates.

X/Y Axis Positions

The machine coordinates of the tool rack position.

Rotary Axis Positions

Carousel/Custom racks only.

Loading Safe Positions X/Y

The machine coordinates of the X/Y loading positions. Side Loading racks only.

Lift Axis Grip

Lift Axis machine coordinate where the tool chuck closes to grip the tool.

Lift Axis Release

Lift Axis machine coordinate where the tool chuck opens to release the tool.

Lift Axis Near Grip

When getting a tool, the Z machine coordinate 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 to use is zero (Z axis completely raised).

Lift Axis Near Release

When putting away a tool, the Lift Axis machine coordinate 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 to use is zero (Z axis completely raised).

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.

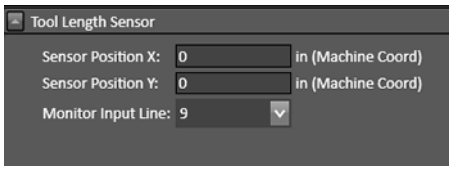
WARNING: IT IS VERY IMPORTANT TO SET THIS VALUE HIGH ENOUGH TO ACCOMMODATE THE LONGEST TOOL. OTHERWISE, A CRASH WILL OCCUR.

Sensing

Tool Length Sensing

Tool Length Sensor

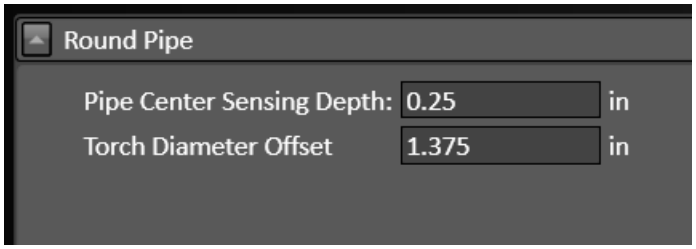
This panel sets parameters used in tool length sensing. **Sensor Position X** and **Sensor Position Y** must be specified in machine coordinates.



Workpiece Sensing

Round Pipe

This panel sets parameters used for sensing round pipe.

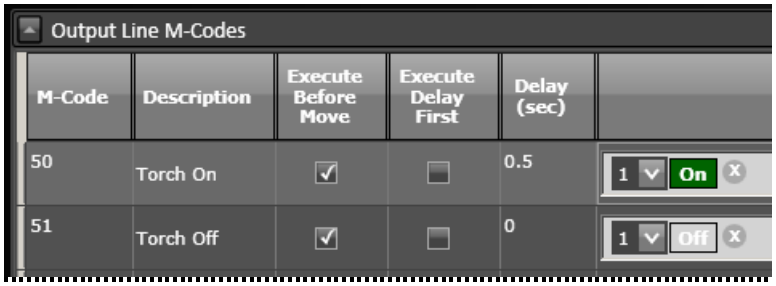


Programming

M-Code Definitions

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.



M-Code

The number of the M-Code that you want to define.

Description

A brief description of the action taken when this M-Code is executed.

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.

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.

Delay (sec)

The duration of the delay that will occur when this M-Code is executed. For example, if the spindle motor takes about 3 seconds to reach full speed, this value should be at least 3.

Output Lines

Each output that is controlled by the M-Code is listed here with the desired state of the output.

Add M-Code Button

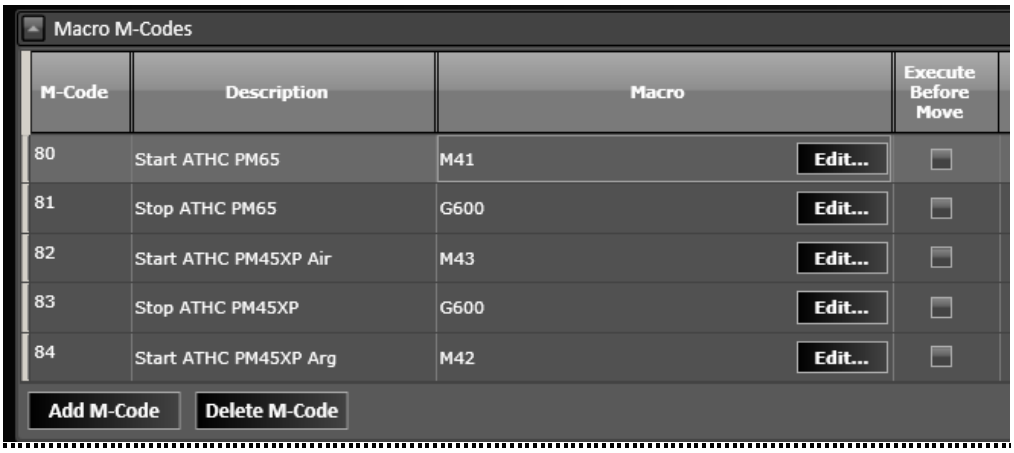
Adds a new line to define an M-Code.

Delete M-Code Button

Deletes the highlighted M-Code definition.

Macro M-Codes

FlashCut can also use M-Codes to call a macro.



M-Code

The number of the M-Code that you want to define.

Description

A brief description of the action taken when this M-Code is executed.

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.

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.

Add M-Code Button

Deletes the highlighted M-Code definition.

Delete M-Code Button

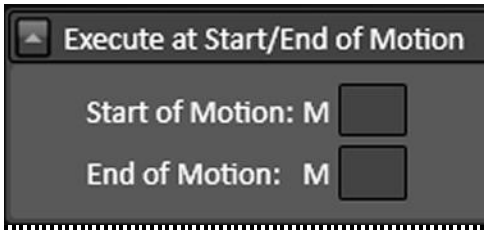
Deletes the highlighted M-Code definition.

M-Code Execution

Any M-Code entered with these configuration types must be defined on the M-Code Definitions panel.

Execute at Start/End of Motion

A typical use for this feature is to turn on a warning light whenever the 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.

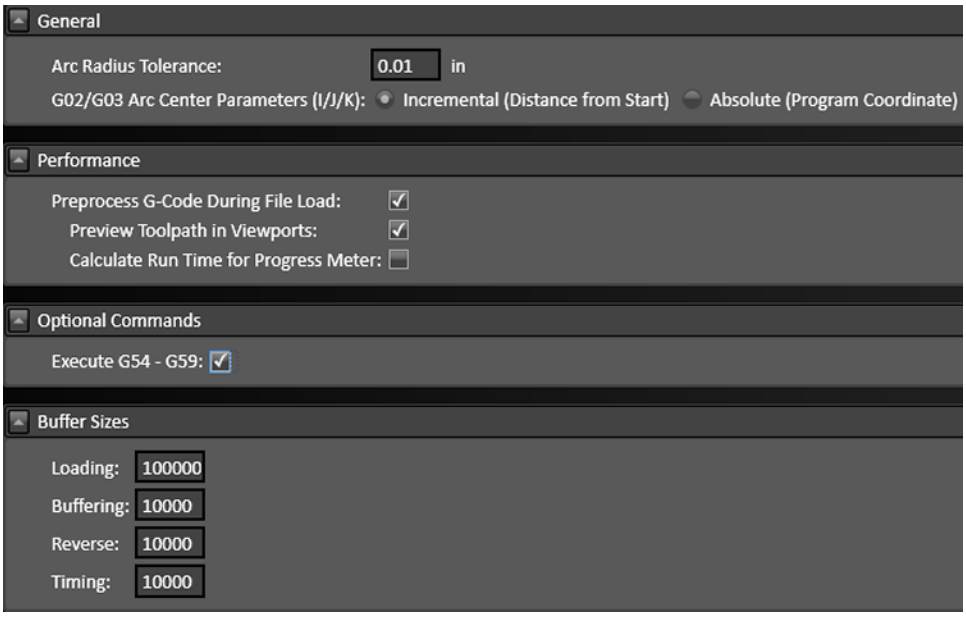
Execute While Connecting/Disconnecting

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.

G-Code



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.

G02/G03 Arc Center Parameters (I/J/K) – Define how arc commands should be interpreted in the g-code program: Incremental or Absolute.

M100/M101 Timeout – Set the timeout period while waiting for an input line state to change for the M100 and M101 commands.

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.

Optional Commands

When this option is selected, G-Code commands G54 through G59 will be executed.

Buffer Sizes

Sets G-Code buffer sizes: Loading, Buffering, Reverse, Timing.

I/O

Input Lines

The Input Line Setup configuration area allows you to configure each of the 32 available input lines.

Set All Wiring

Sets the **Wiring** option for every line to either **Normally Open** or **Normally Closed**.

Line Setup

You may set the following options for each input line.

In addition, you may click one of the two buttons to set each input line as normally open or normally closed.

Line

Identifies the line number on the Signal Generator's Input connector.

Description

A brief explanation of the input line, used by FlashCut in various system messages.

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.

Axis

Identifies the location (by axis) for all limit and home switches.

Sub-axis

If your machine uses two motors to drive an axis (defined on the Machine>Axes panel), FlashCut will include two sub-axes in the list of choices.

Position

Identifies the direction, for a given axis, for all limit and home switches.

Sensing Debounce

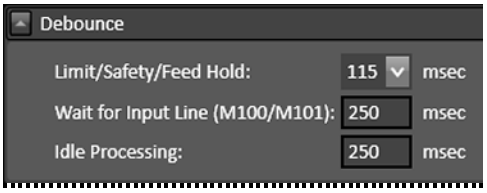
For a switch to successfully reach a desired state, it must hold that state continuously for the debounce duration (in seconds).

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.

Debounce

This is the duration that a switch must be continuously tripped for FlashCut to recognize it as tripped.\

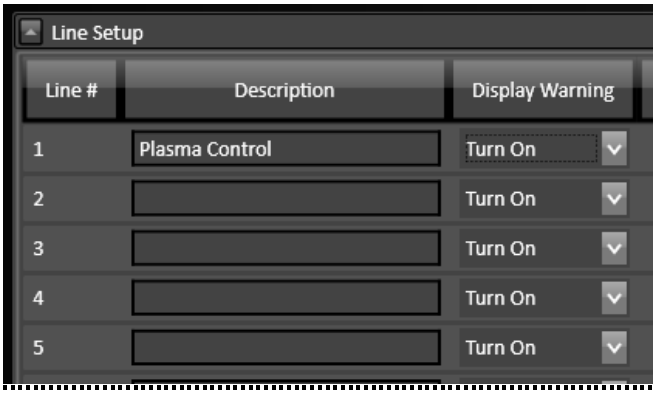


Limit/Safety/Feed Hold – Sets the duration in milliseconds for these switch functions.

Wait for Input Line – Sets the duration in milliseconds of the debounce for operations M100 and M101.

Idle Processing – Sets the duration in milliseconds.

Output Lines



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**.

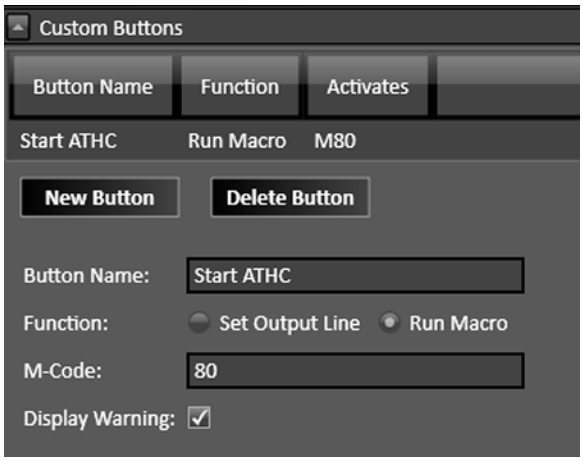
User Interface

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).

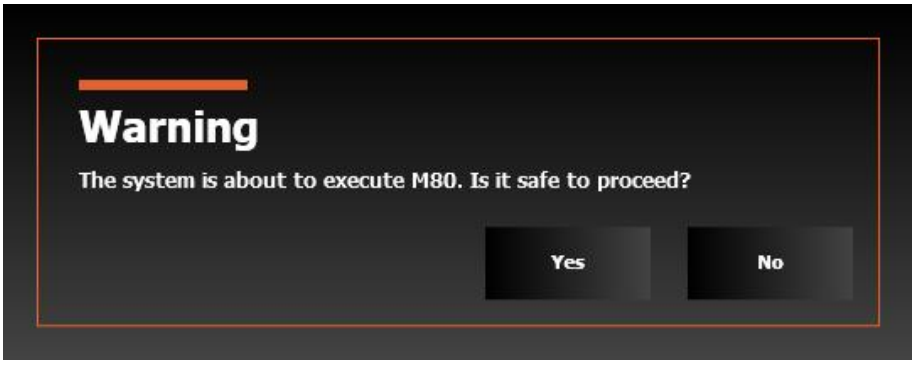
Note that the M-Code has already been defined, and that the Display Warning checkbox has been selected.



Custom buttons appear in the CNC window. They are located in a panel to the left of the drawing workspace.

The button from the 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.

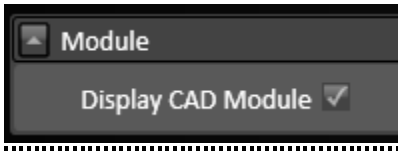


CAD

General

Display CAD Module

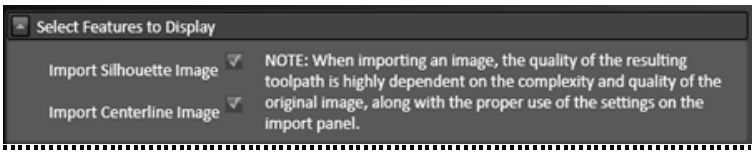
When selected, the CAD module will be available in FlashCut.



Features

Select Features to Display

The Select Features to Display configuration allows you to select the following items to display:



Import Silhouette Image

This enables the use of the silhouette image import in the CAD module.

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.

Use Layers

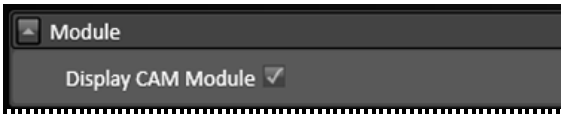
This enables the manipulation of Layers in the CAD module.

CAM

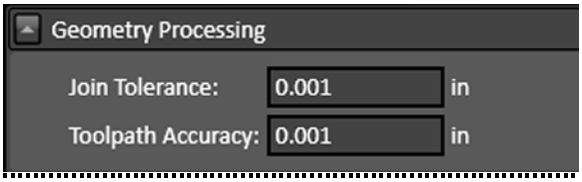
General

Module

When selected the CAM module will be available in FlashCut.



Geometry Processing



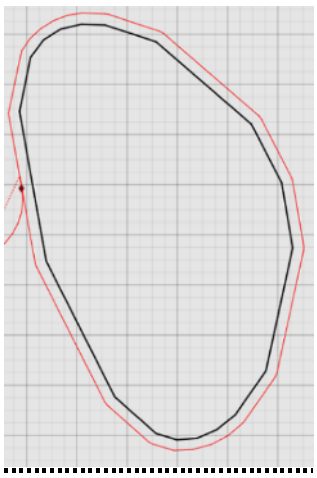
Join Tolerance

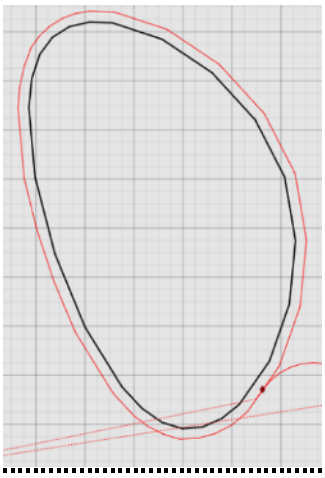
Defines tolerance in system units.

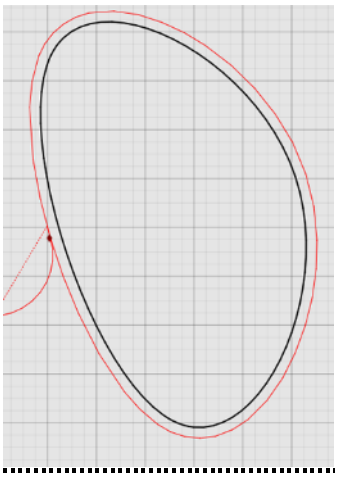
Toolpath Accuracy

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 levels of toolpath accuracy: 0.1, 0.01, and 0.001 (left to right).







Pipe Beveling

Enable the use of toolpath smoothing for pipe beveling (recommended).

Features

Select Features to Display

Grid Nesting

This enables the use of the Grid Nesting tool in the CAM module.

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.

Server Address and Path

Enter the URI for the server including the sub-folder paths.

Username

Enter the username for the FTP account in the Username field.

Password

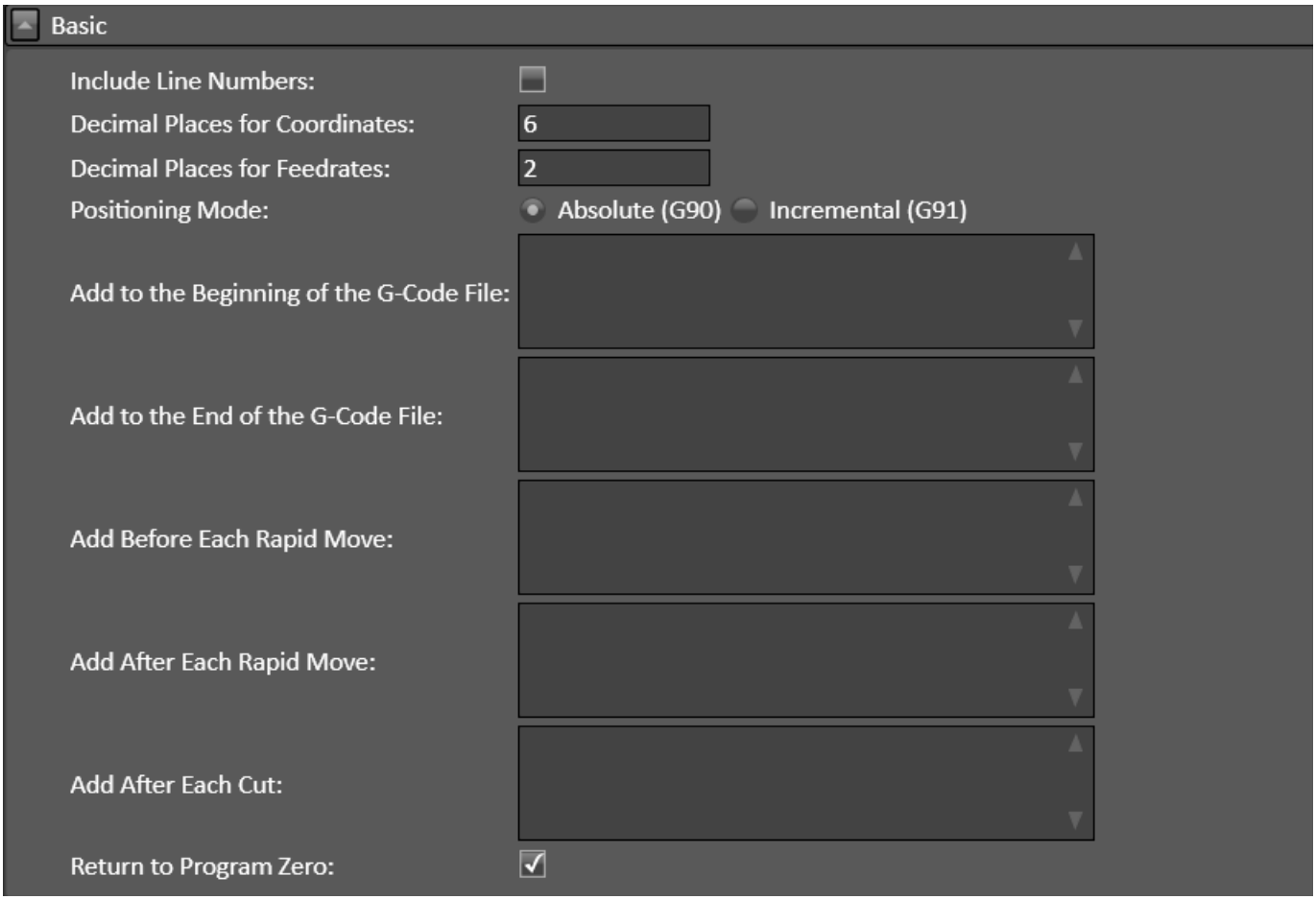
Enter the password for the FTP account in the Password field. FTP account passwords are typically case-sensitive.

Use FTP Upload Panel

If enabled, an FTP upload panel appears in the parameters pane with various control options. If not enabled, the file is uploaded with a default filename (e.g., gcode.nc).

Post Processor

Basic Post Processor Settings



Include Line Numbers

Check this box if you want to number all of the G-Code lines in the program that's being created.

Decimal Places for Coordinates

Enter the number of decimal places to be used for coordinates.

Decimal Places for Feedrates

Enter the number of decimal places to be used for feedrates.

Positioning Mode

Select either absolute or incremental.

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.

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.

Add Before Each Rapid Move

Any G-Code command entered here will be added **before** all rapid moves (G00).

Add After Each Rapid Move

Any G-Code command entered here will be added **after** all rapid moves (G00).

Add After Each Cut

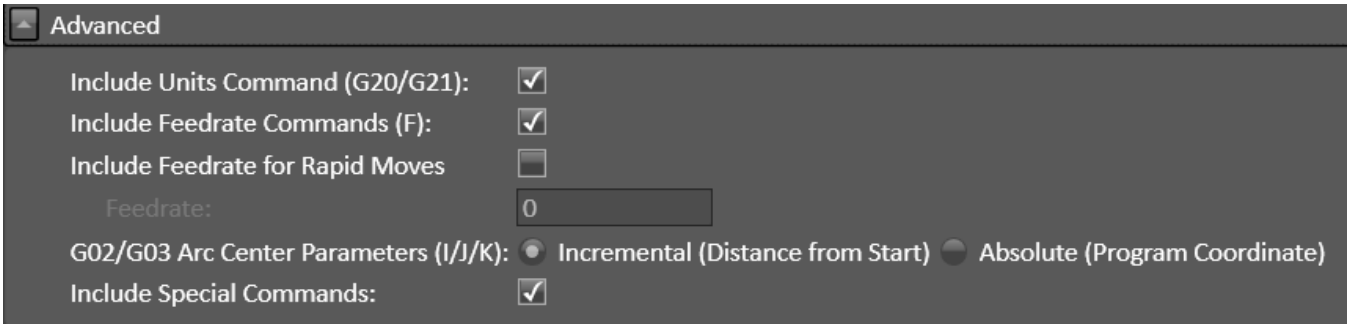
Any G-Code command entered here will be added **after** each cut.

Return to Program Zero

When selected, this feature is enabled.

Advanced Post Processor Settings

These settings should only be modified from the default recommended values as necessary when using a 3rd party CNC application to run the G-Code program.



Include Units Command

This is enabled by default. This will 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.

Include Feedrate Commands

This is enabled by default. This will include the F feedrate command for feedrate moves in the G-Code program.

Include Feedrate for Rapid Moves

This is disabled by default. This will include the **F** feedrate command for rapid moves in the G-Code program.

G02/G03 Arc Center Parameters (I/J/K)

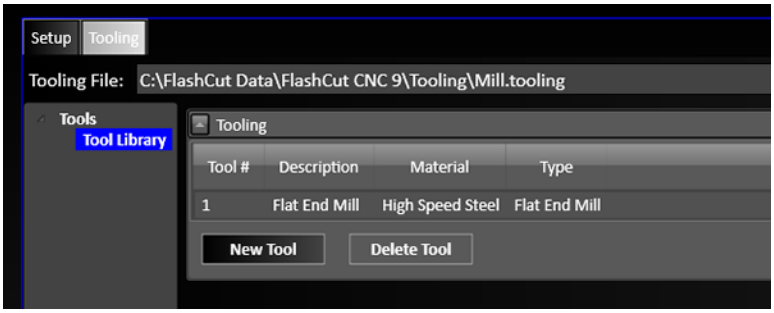
This is set to Incremental by default. This determines how the clockwise (G02) and counterclockwise (G03) arc commands will be generated in the G-Code program. Select either incremental (distance from start), or absolute (program coordinates).

Include Special Commands

This is enabled by default. When enabled, #FabHead commands, #Workpiece definitions, #Process settings, and #File properties will be included in the G-Code file. For plasma fabrication heads configured with THC, commands to enable and disable torch height control will be included in the G-Code file.

Tooling Configuration Tab

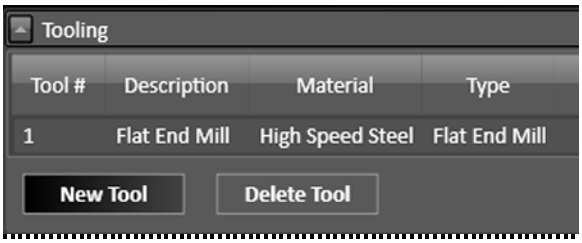
When the tab is selected, the tooling file setup window and Tool Library are visible.



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.

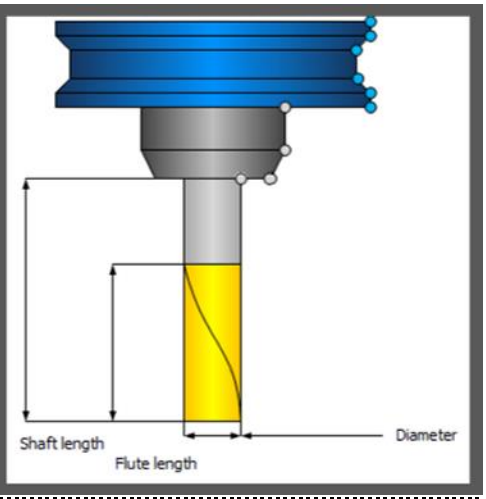
Type – The following types of tools can be defined: Ball End Mill, Drill, Flat End Mill, Round Edge End Mill, Tapered Edge End Mill, Creasing Wheel, Dual Sided Knife, Rotary Knife, and Straight Knife.

To add a tool to the library, select **New Tool**. To remove a tool from the library, select **Delete Tool**.

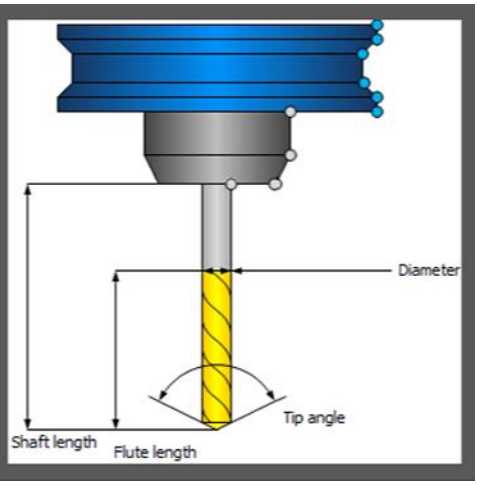
When adding a tool, you must define its parameters before it can be used.

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 units.

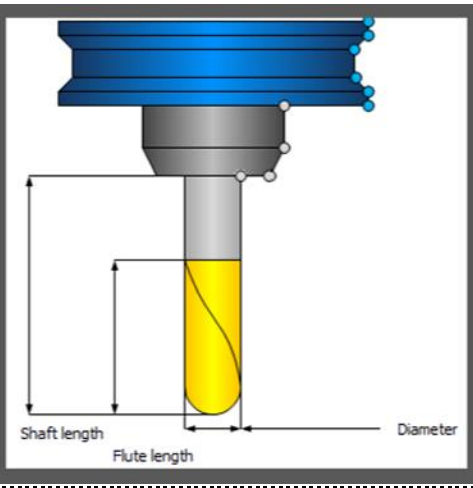
Tool Number:	1	
Tool Offset:	0	in
System Units:	English	
Description:	Flat End Mill	
Material:	High Speed Steel	
Nominal Diameter:	0.2500	in
Flute Length:	0.2500	in
Shaft Length:	2.0000	in
# Of Flutes:	3	
Cutting Direction:	CW	



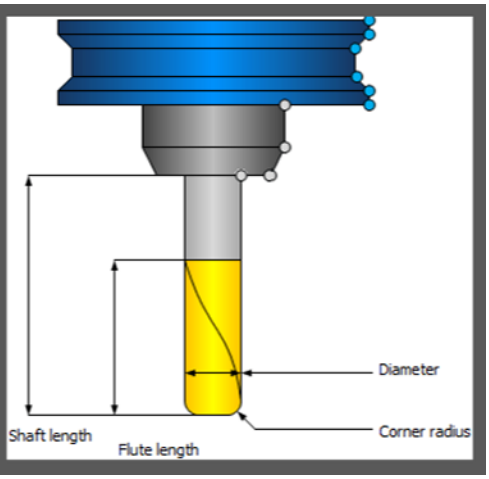
Tool Number:	3	
Tool Offset:	0	in
System Units:	English	
Description:	Drill	
Material:	Carbon Steel	
Nominal Diameter:	0.3125	in
Flute Length:	1.9000	in
Shaft Length:	3.7500	in
# Of Flutes:	2	
Cutting Direction:	CW	
Tip Angle:	118	°



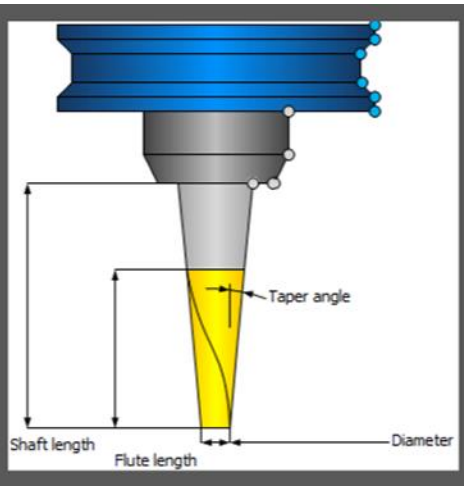
Tool Number:	2	
Tool Offset:	0	in
System Units:	English	
Description:	Ball End Mill	
Material:	Uncoated Carbide	
Nominal Diameter:	0.5000	in
Flute Length:	2.0000	in
Shaft Length:	4.0000	in
# Of Flutes:	2	
Cutting Direction:	CW	



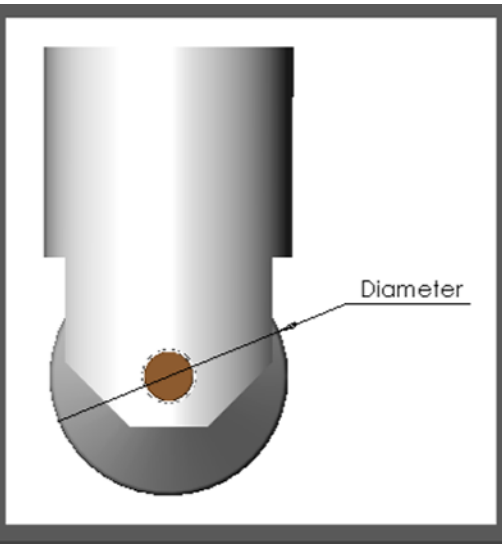
Tool Number:	4	
Tool Offset:	0	in
System Units:	English	
Description:	Round Edge End Mill	
Material:	Uncoated Carbide	
Nominal Diameter:	0.3125	in
Flute Length:	0.8125	in
Shaft Length:	2.5000	in
# Of Flutes:	4	
Cutting Direction:	CW	
Corner Radius:	0.045	in



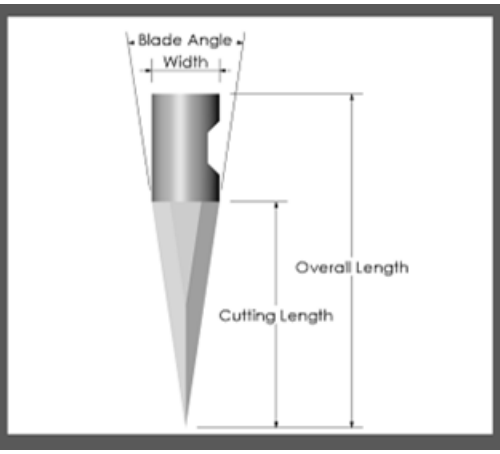
Tool Number:	5	
Tool Offset:	0	in
System Units:	English	
Description:	Tapered Edge End Mill	
Material:	High Speed Steel	
Nominal Diameter:	0.3750	in
Flute Length:	0.5000	in
Shaft Length:	2.5000	in
# Of Flutes:	3	
Cutting Direction:	CW	
Angle:	20	°



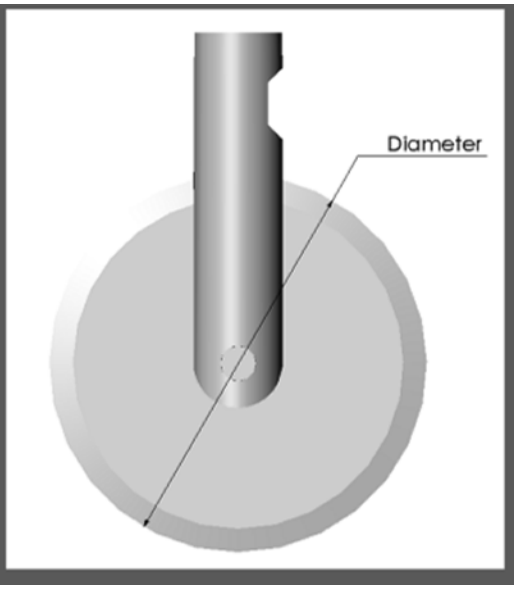
Tool Number:	6	
Tool Offset:	0	in
System Units:	English	
Description:	Creasing Wheel	
Diameter:	2	in



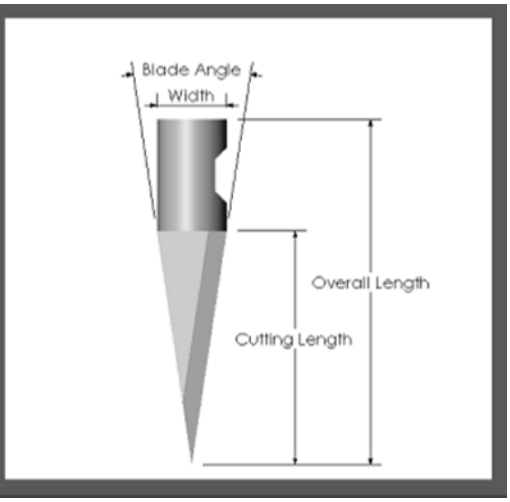
Tool Number:	8	
Tool Offset:	0	in
System Units:	English	
Description:	Dual Sided Knife	
Cutting Length:	2	in
Width:	0.25	in
Blade Angle:	7.16	°
Overall Length:	4	in



Tool Number:	7	
Tool Offset:	0	in
System Units:	English	
Description:	Rotary Knife	
Diameter:	1	in



Tool Number:	9	
Tool Offset:	0	in
System Units:	English	
Description:	Straight Knife	
Cutting Length:	2	in
Width:	0.25	in
Blade Angle:	7.16	°
Overall Length:	4	in



CAD/CAM Common Interface Elements

Menu Bar

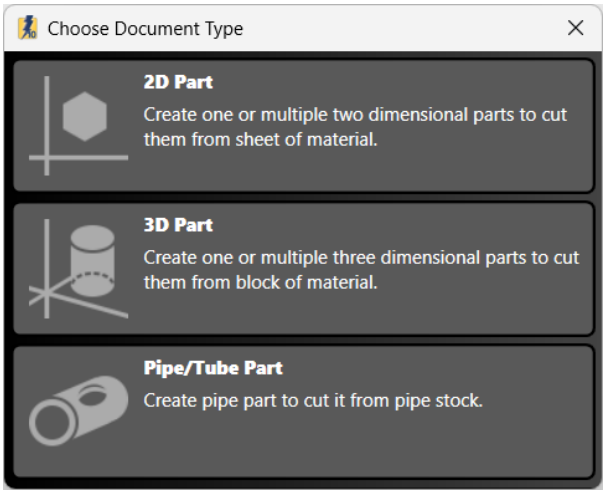
The menu bar is the same in both **FlashCut CAD** and **FlashCut CAM**. The menu bar contains menu items for system commands. Hotkey commands are listed next to their respective functions. Accelerator keys are displayed when you press the **{Alt}** key.

File Menu

New (Ctrl+N)

Creates a new FlashCut CAD/CAM document.

Depending on the features available on your system, you may see a prompt to Choose Document Type: **2D Part**, **3D Part**, or **Pipe/Tube Part**.



Open (Ctrl+O)

Opens an existing FlashCut CAD/CAM document. The documents have a .cadcam file extension. Note: Documents saved in earlier versions of FlashCut can be opened, but documents saved in newer versions of FlashCut may not be compatible.

Close

Closes the current CAD/CAM document. If the document is unsaved, the software prompts you to save the document before closing.

Close All

Closes all CAD/CAM documents currently open. If any document is unsaved, the software prompts you to save the document before closing.

Save (Ctrl+S)

Saves the current CAD/CAM document using the existing file name and location. It will not save any other CAD/CAM documents that are open. FlashCut will prompt you to create a file name and destination. FlashCut saves documents in the FlashCut CAD/CAM file format with a .cadcam file extension. These documents are readable by FlashCut CAD/CAM software. The Save command is unavailable when no recent changes have been made.

Save As...

Saves the current document to a new file name or destination. It will not save any other CAD/CAM documents that are open. FlashCut prompts you to create a file name and destination. Saved documents use the extension .cadcam. These documents are readable by FlashCut CAD/CAM software.

Save All

Saves all open documents, not only the current document that is being viewed. This command is unavailable when no recent changes have been made.

Export

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

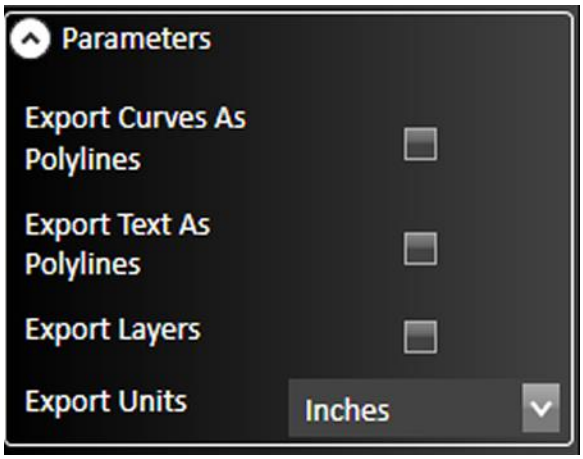
Both commands bring up a Windows dialog box that allows you to select the location, filename, and specific file format.

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 file Save As dialog then provides a configuration panel in the **Parameters area**. The content of the drawing will determine which parameters are available. After setting the parameters for the file, you may either **Apply** 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 Layers

When selected, layer information will be saved in the DXF file.

Export Units

Choose from either inches or millimeters.

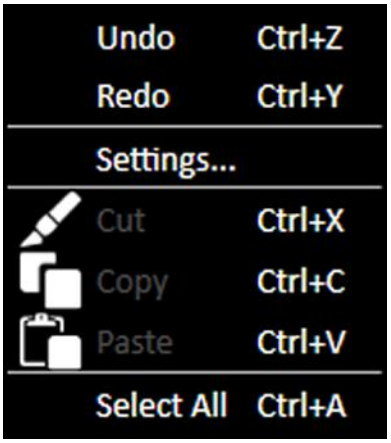
Print

In CAM, depending on the features available on your system, you may have a Print menu. This provides the option to **Print Labels** or **Print Nest** that are described later in this document.

Exit

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

Edit Menu



Undo (Ctrl+Z)

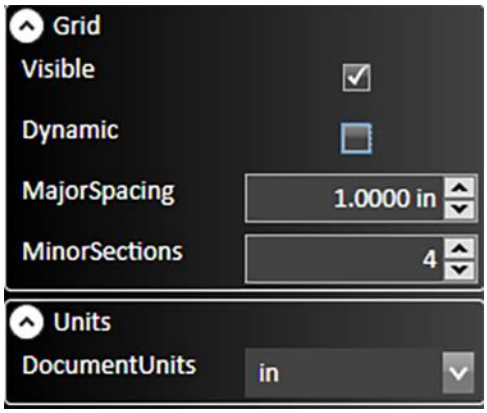
Reverses the previous drawing action. Up to 20 actions can be reversed.

Redo (Ctrl+Y)

Repeats the previous drawing action or reverses the Undo stack.

Settings...

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



Grid

Visible

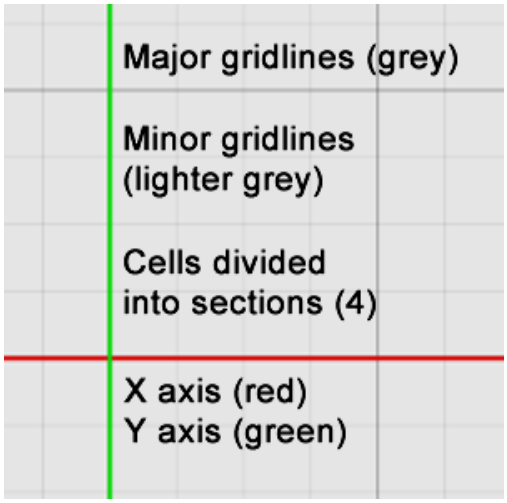
This option will hide/display the gridlines in the workspace.

Dynamic Grid

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

Major Spacing

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



Minor Sections

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

Units

Document Units

Select the units of the CAD/CAM document; mm (millimeters) or in (inches).

FlashCut gives you the option to convert any existing parts (drawing objects) in the workspace. Selecting **Yes** rescales the existing values into new units (i.e., a 1 inch circle is a 25.4mm circle). Selecting **No** reinterprets the values into new units (i.e., a 1 inch circle becomes a 1mm circle).

Note that changing the units for a drawing will clear all CAM data.

Cut (Ctrl+X)

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

Note that objects are selected by clicking on them with the selection arrow (which becomes available by pressing the **Esc** key). To select all segments of a feature (**chain select**), hold down the **Alt** key.

You can include other objects by holding down the **Ctrl** key while you select the objects that you want. You may also select multiple objects by creating a selection box with the selection arrow. Creating a selection box that goes from left to right will select all objects that it touches and a selection box that goes from right to left will select all objects that it completely envelopes.

Copy (Ctrl+C)

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

Paste (Ctrl+V)

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

Select All (Ctrl+A)

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

Document Tabs

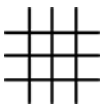
Tabs for all open documents are displayed in the upper left corner of the CAD/CAM drawing workspace. Clicking the "+" button to the right of the tabs creates a new document.

Part Shading and Grid Tools

These tools are displayed in the upper right corner of the CAD/CAM drawing workspace.

Toggles **part shading** in all open documents.

Toggles the **grid** in the active drawing workspace.



Parameters Area

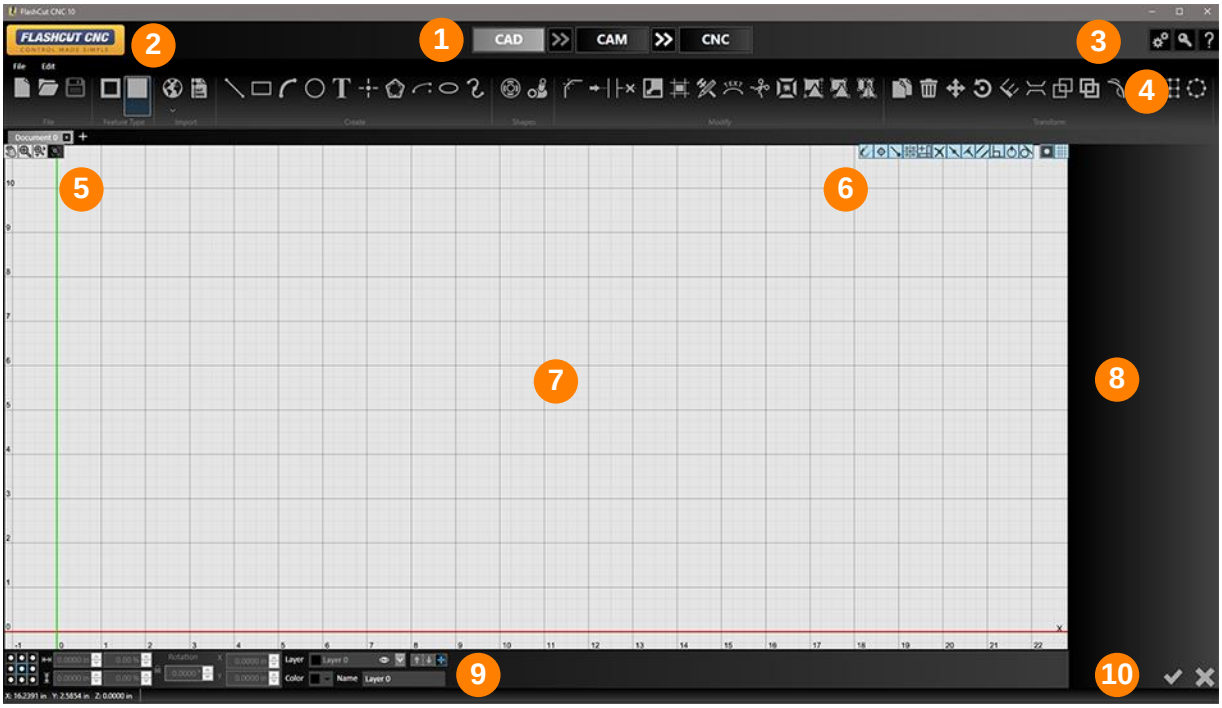
Parameters for selected tools or objects are shown in this area in both **FlashCut CAD** and **FlashCut CAM**. Parameters displayed in this area will be described in the sections of this document where the tools or objects are discussed.

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.

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.



1. Navigating Through CAD CAM CNC
2. Menu
3. Configuration / License / Help
4. CAM Ribbon
5. Document Tabs
   Pan and Zoom Tools
6. Display Options
7. Viewport for CAD
8. Parameters Area
9. Manipulation Tools
10. Apply / Cancel

CAD Ribbon

The ribbon features an assortment of command icons to create, modify, and transform objects in the drawing window. The ribbon will show features based on the current document type (2D Part, 3D Part, Pipe)

File

New Document (Ctrl+N)

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

Open... (Ctrl+O)

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

Save (Ctrl+S)

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

Feature Type

This is for 2D Part and Pipe Documents. 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 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.

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 in CAM. These options will be greyed out in the ribbon.

Make Top Level Feature a Part

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

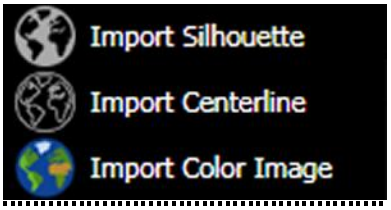
Import Tools

The import tools are used for adding images or DXF CAD drawings from an external file to the drawing.

Import Silhouette, Centerline, or Color Image

Selecting the **Import** icon from the ribbon brings up a menu with three different options: **Import Silhouette Image**, **Import Centerline Image**, and **Import Color Image**.

FlashCut accepts numerous image file types: *.bmp *.dib *.jpg *.jpeg *.jpe *.png *.pbm*.pgm *.ppm *.sr *.ras *.tiff *.tif



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

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

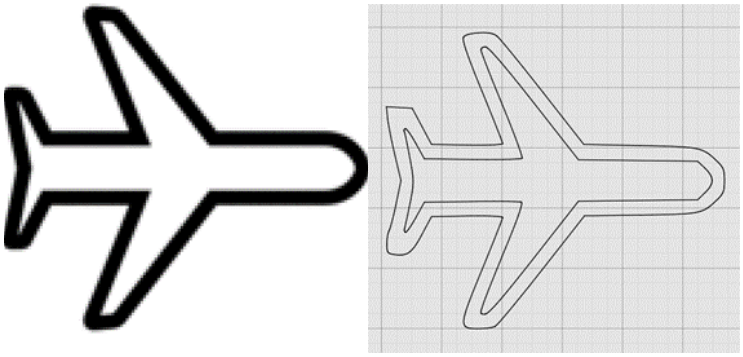
Import Color Image creates an outline based on the colored areas of an image.

Understanding silhouette and centerline images

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

Import Silhouette Image

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



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

Choose Image

Click **Browse...** to select an image file from the computer. Select **Open** to call up the desired image. A list of file formats that can be imported appears on page 108.

Scale

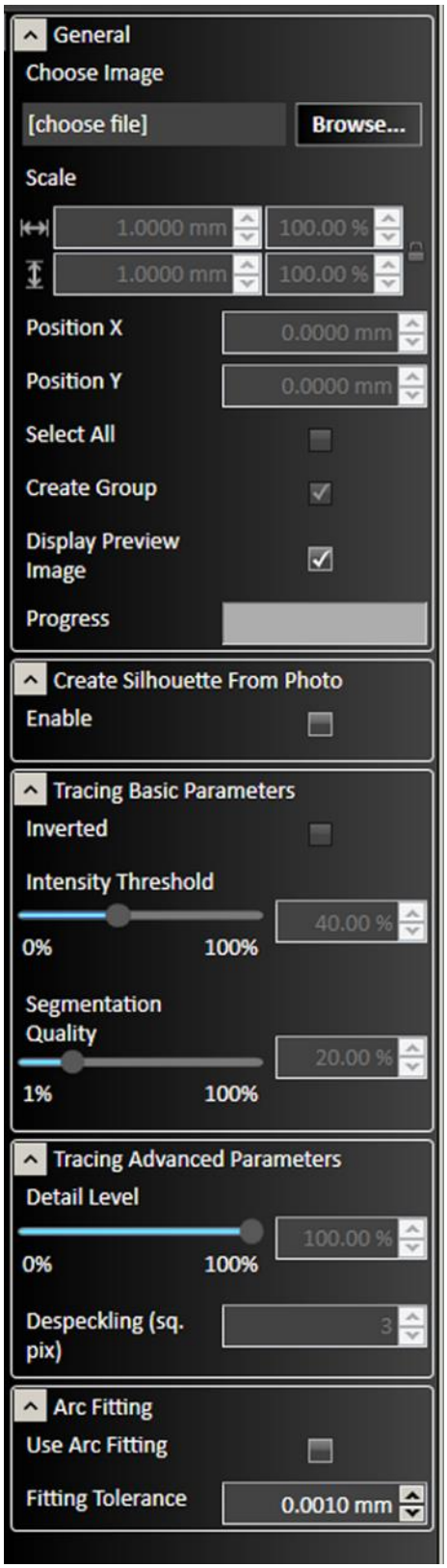
Select how large or small the image will appear in the drawing relative to its original size. Scaling can be done as an absolute dimension or as a percentage of the original image size.

Position X

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

Position Y

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



Select All

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

Create Group

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

Display Preview Image

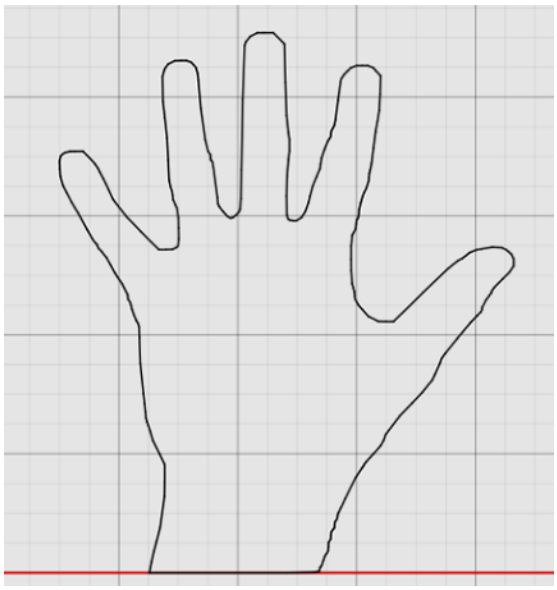
Superimposes a translucent preview of the imported image while the new drawing is being generated.

Progress

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

Create Silhouette from Photo

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



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

Background color –Select which corner of the image that is to be used to sample the background color. The color sampled from the specified corner will be set as the background color. When importing a photo, the edges of the object are determined by the color contrast between an object and the selected background color. The **Color Tolerance** setting is used to adjust the level of contrast.

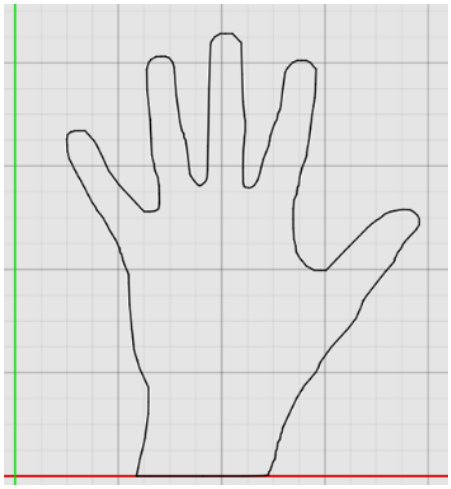
Pick small details – Increase the value if the algorithm missed some small details. Import performance may be lowered if this value is increased. 10% is a good default value.

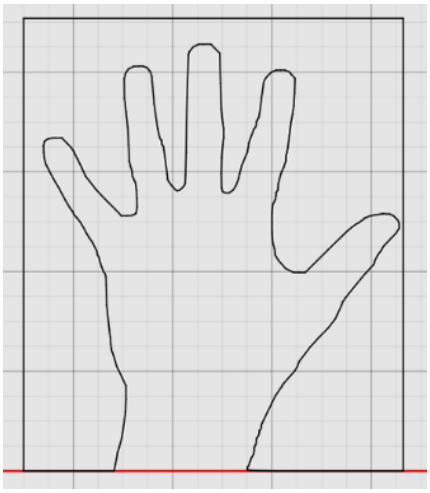
Color Tolerance – This tolerance is used to determine the contrast between the background and the silhouette. Pixels within this range will be considered part of the background. Range: 0-100.

Tracing Basic Parameters

Inverted

When checked, this option reverses the shapes enclosed by the curves of the silhouette. 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.

Tracing Advanced Parameters

Detail level

Select the degree of accuracy of the lines in the silhouette to the original image. A smaller detail level allows for more variation from the original drawing, while a larger level replicates the image more closely. Range: 0-100.

Despeckling

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

Arc Fitting

Use Arc Fitting

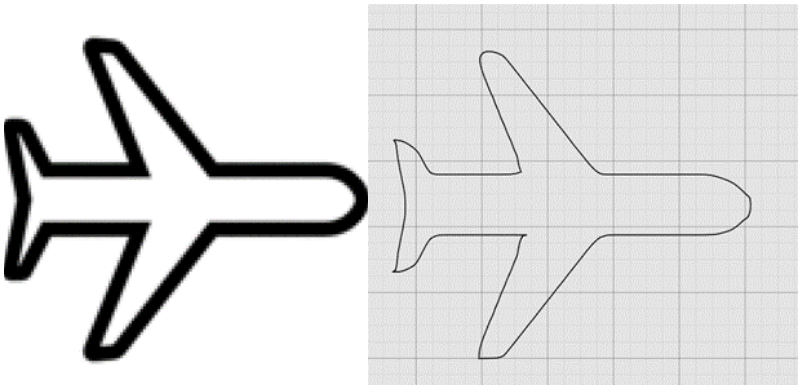
When selected, arc fitting will be applied to the imported image.

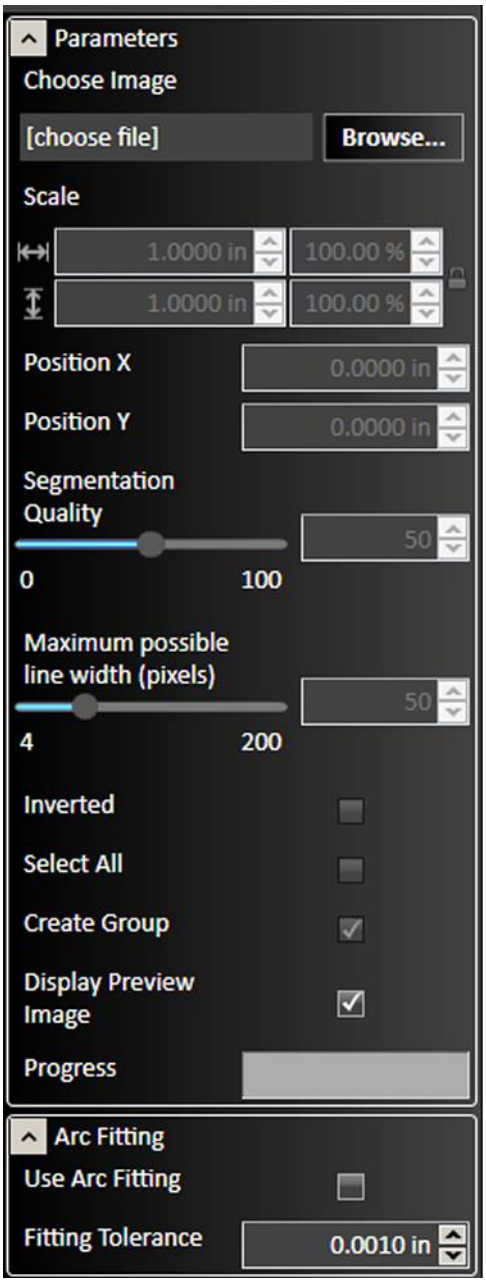
Fitting Tolerance

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

Import Centerline Image

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





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

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 108.

Scale

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

Position X

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

Position Y

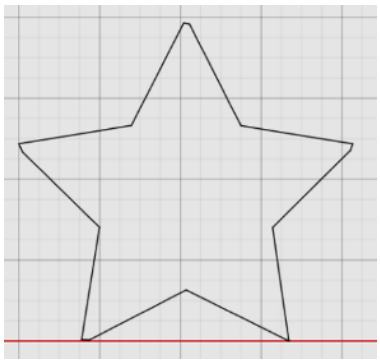
Enter the value for the Y coordinate of the lower left corner of the imported image.

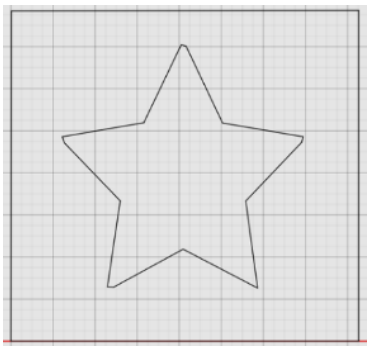
Segmentation Quality

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

Inverted

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





Select All

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

Create Group

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

Display Preview Image

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

Progress

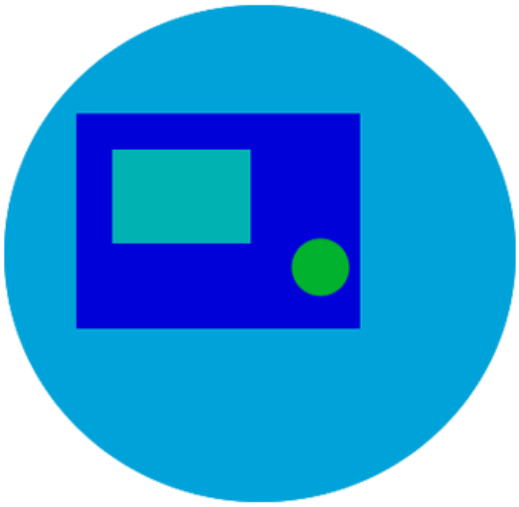
Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for the rendering to finish between edits.

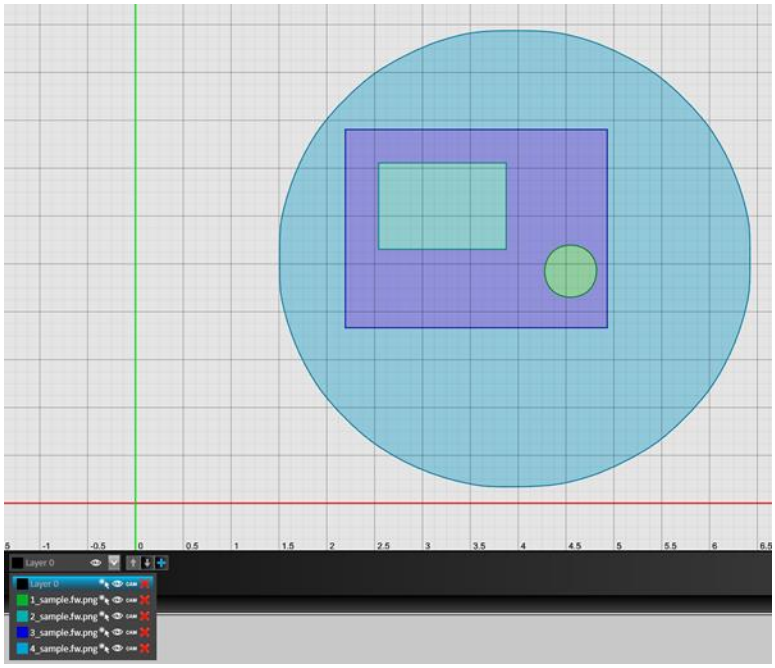
Arc Fitting

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

Import Color Image

This tool creates shapes from the source drawing based on color. Boundaries between different colors define where the lines are drawn. Each new shape is placed on a unique layer, which is color coded. In the example below, the original bitmap image (left) is translated into four shapes (right), each occupying its own color-coded layer. Refer to the Layers section of this document for more information about displaying and manipulating drawing objects using layers.





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

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 108.

Scale

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

Position X

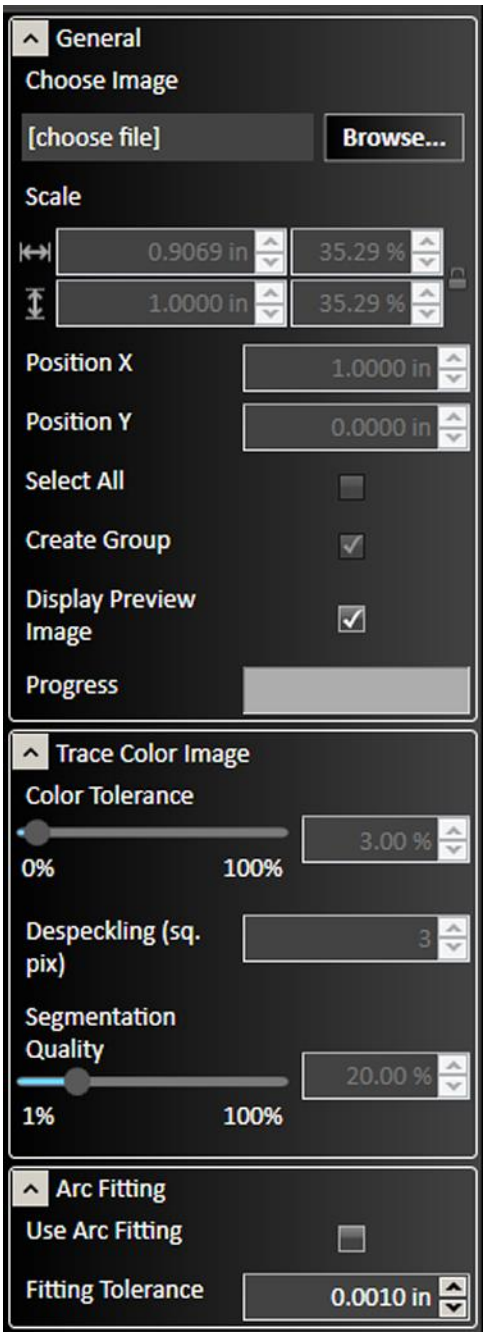
Enter the value for the X coordinate of the lower left corner of the imported image.

Position Y

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

Select All

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



Create Group

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

Display Preview Image

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

Progress

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

Color Tolerance

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

Despeckling

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

Segmentation quality

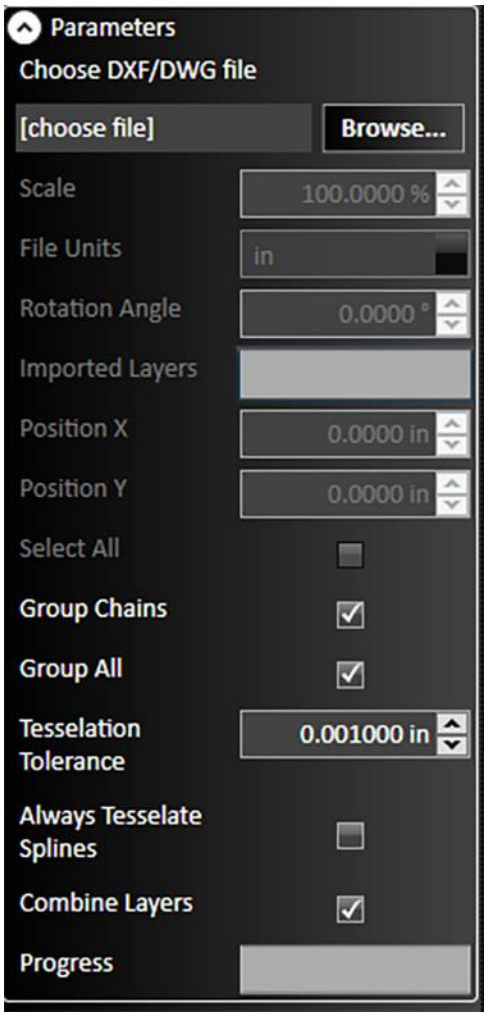
Select how finely the program will divide curves. FlashCut CAD automatically breaks curves into separate line segments. When segmentation quality is increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing. Range 1-100.

Arc Fitting

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

Import DXF/DWG File

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



Choose File

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

Scale

Select how large or small the image will appear in the drawing relative to its original size. X and Y values will be scaled equally.

File Units

Select units used in the document.

Rotation Angle

Set rotation angle in degrees.

Imported Layers

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

Position X

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

Position Y

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

Select All

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

Group Chains

When checked, chains in the imported drawing will be grouped.

Group All

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

Tessellation Tolerance

This value is for entities that cannot be directly imported into FlashCut.

Always Tessellate Splines

When checked, all splines will be tessellated.

Combine Layers

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

Progress

Shows progress of file import.

Create

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.

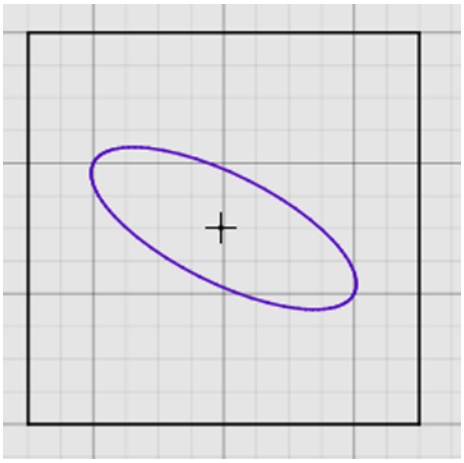
After changes are complete, you may either **Apply** or **Cancel** the changes.

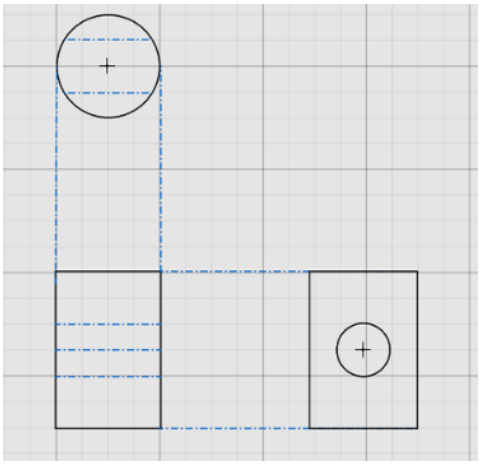
Each tool (except for Point) lets you define the **Treatment** parameter to determine whether the element is cut or only marked.

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 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 the **Create** tools is influenced by the active **Snap** tools.

Line

Lines may be created either as **Continuous Lines** by adding segments and vertices with each mouse click, or as a simple **Two Point Line**.

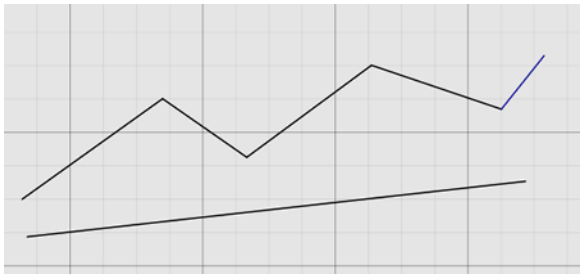
When creating a Two Point Line, the properties may be adjusted manually in the parameters window. Any single line segment (e.g., any segment that is part of a polygon or a multi-segment line) may be selected and its properties displayed.

Creation mode

Continuous Line to create a series of lines.

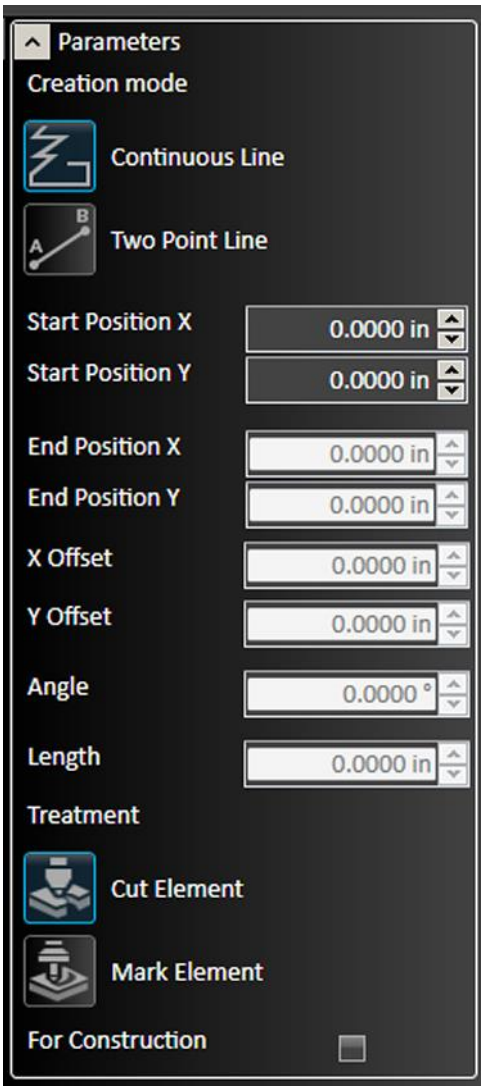
Two Point Line to create one line.

Examples of each type are shown below.



Position, offset, angle and length

Specify the attributes of a specific line segment.



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.

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

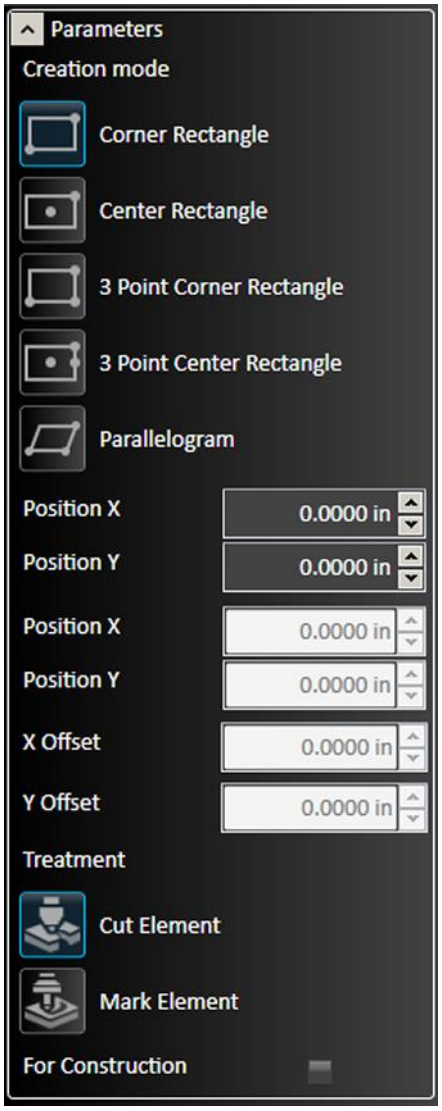
The first two points clicked set the length of the center line. Drag to set the distance from the center line to the lines parallel to it.

Parallelogram

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

Position, offset, midpoint, corner

Specify the attributes of a specific point.



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.

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

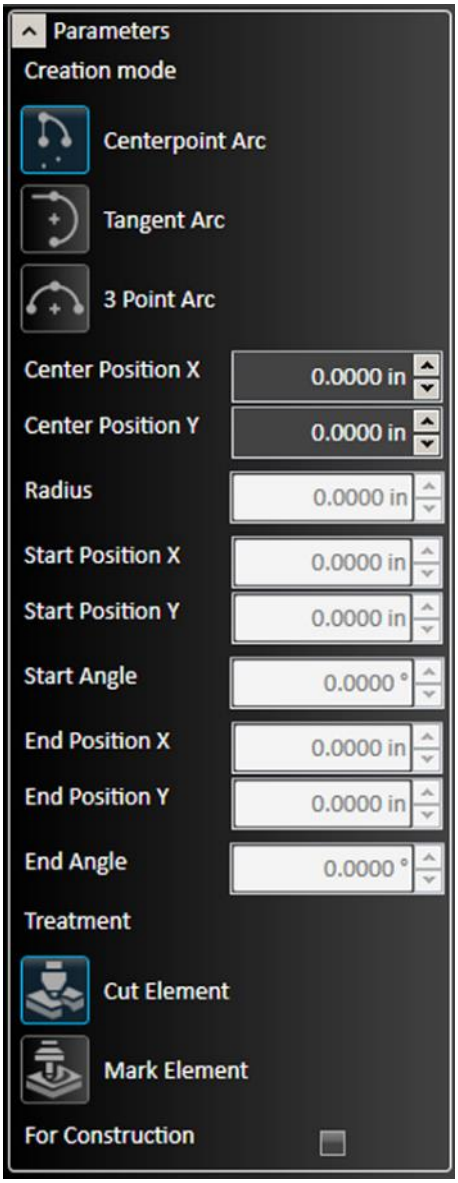
Click on the end point of an existing entity (point, line, rectangle, etc.). Drag to set the end point of the arc.

3 Point Arc

Click and drag to set each point.

Positions, offsets, angles, and/or radius

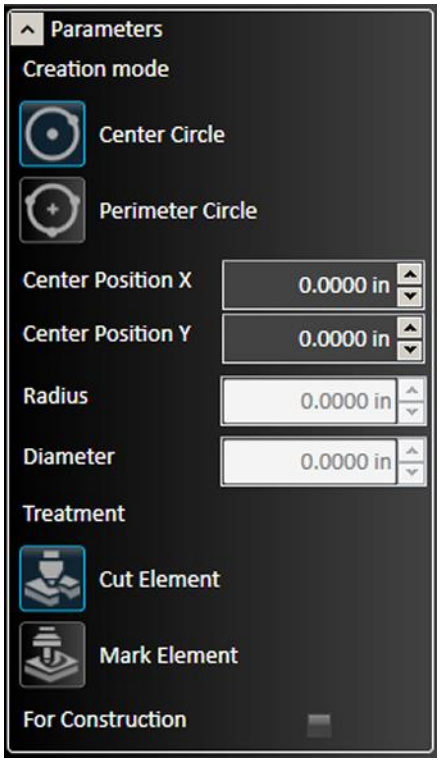
Specify the attributes of the arc.



Circle

FlashCut CNC offers two different options for constructing circles. The **Center Circle** enables you to select a center point, followed by a point on the circumference of the circle.

The **Perimeter Circle** prompts you to select three points on the circumference, through which the circle will be drawn.



Creation Mode

Select the 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 points that define the perimeter of the circle.

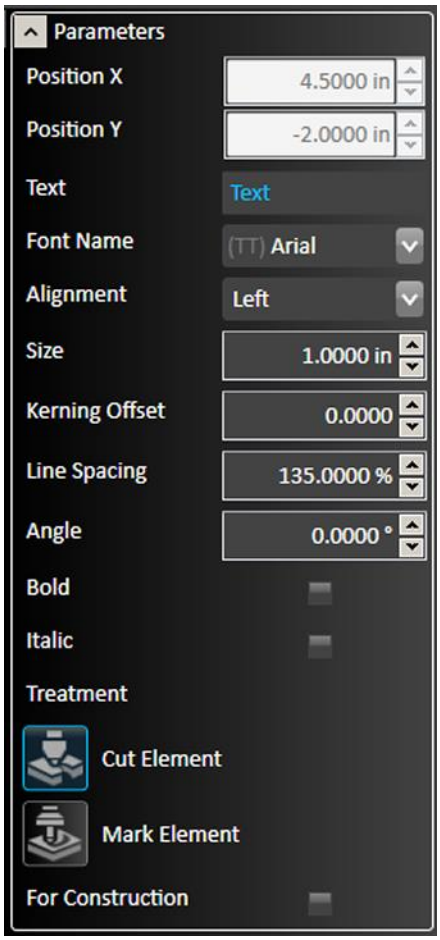
Position, radius, and diameter

Specify the attributes of the circle.

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.



Position X and Position Y

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

Text

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

Font Name

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

Alignment

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

Size

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

Kerning Offset

Specify the spacing between letters.

Line Spacing

Select a value between 50% and 300%.

Angle

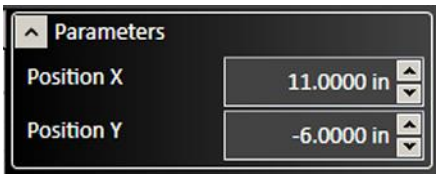
Specify the angle relative to the anchor point. 360 degree rotation in either direction permitted.

Bold and Italic

Check box to apply style.

Point

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

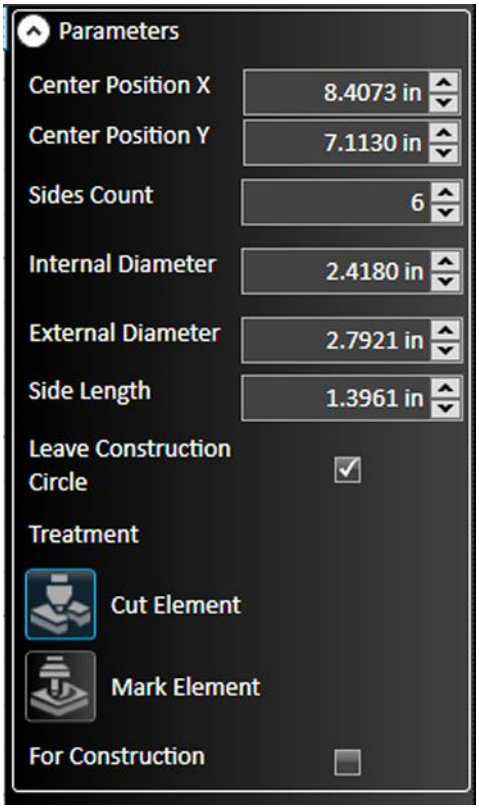


Position X and Position Y

Specify the position of the point relative to the origin.

Polygon

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



Center X/Y Position

Specify the center of the polygon relative to the origin.

Sides Count

Defines the type of polygon. The minimum number of sides is three.

Internal Diameter

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

External Diameter

The diameter of an exterior circle tangent to all vertices.

Side Length

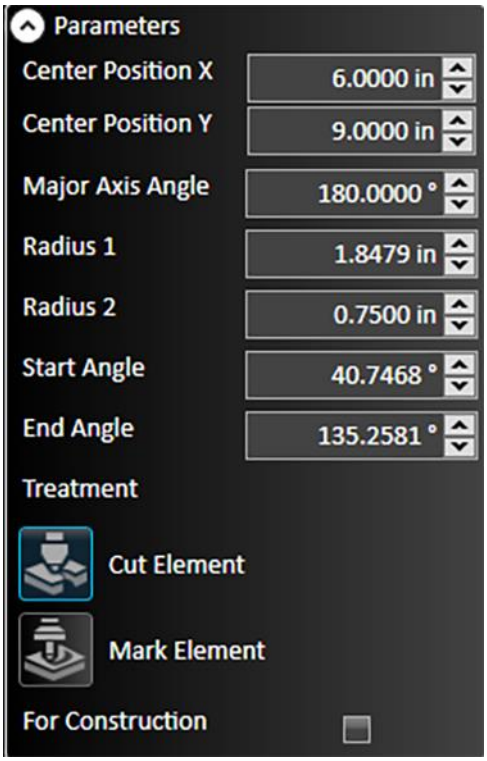
Length of each side.

Leave Construction Circle

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

Elliptic Arc

Creates an arc that is a section of an ellipse.



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.

Finally, select a start point and then an end point to the arc.

Center Position X and Y

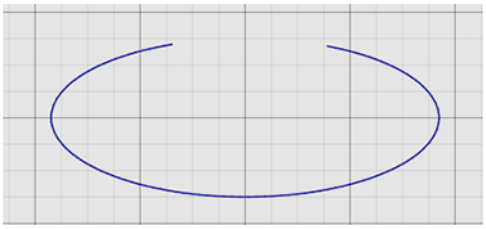
Specify the coordinates of each center point.

Major Axis Angle

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

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.

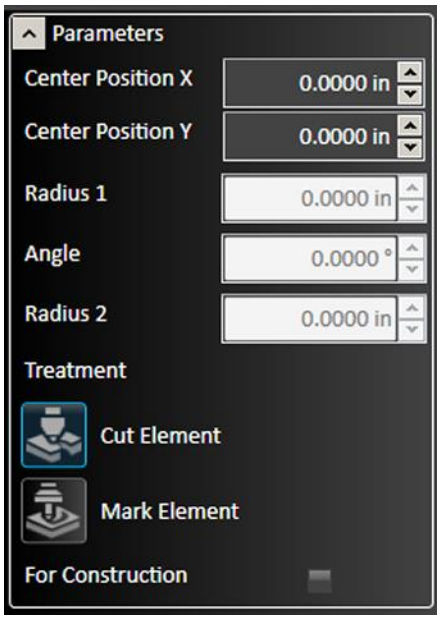


Start and End Angle

Define the beginning and end points of the arc relative to the center point.

Ellipse

Creates an ellipse.



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.

Center Position X and Center Position Y

Specify the coordinates of each center point.

Radius 1 and Radius 2

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

Angle

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

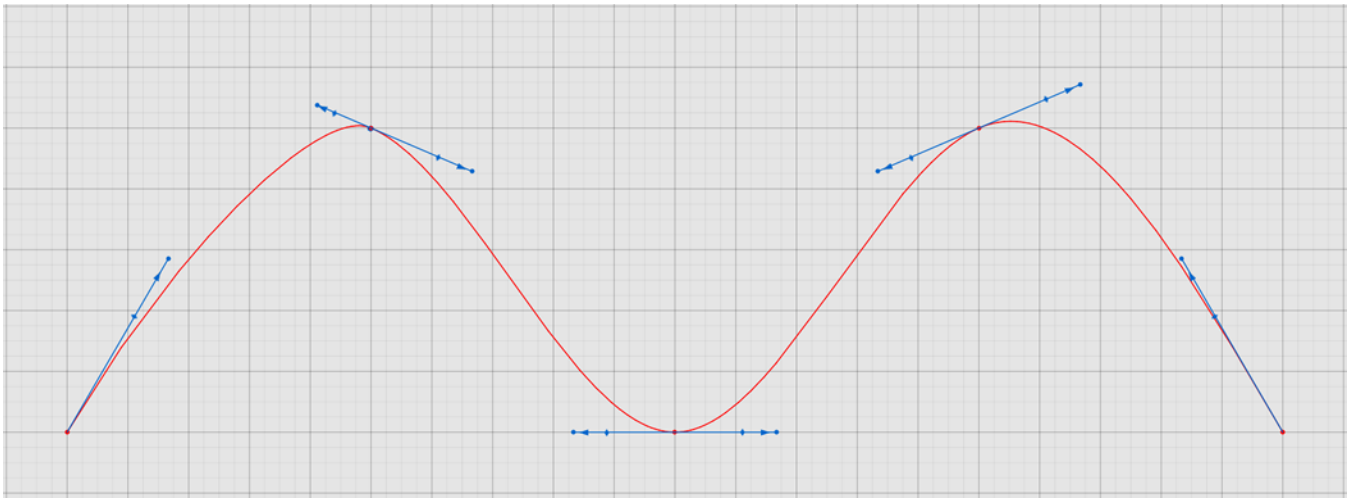
Spline

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.

Drawing each curve segment

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



Position X and Position Y

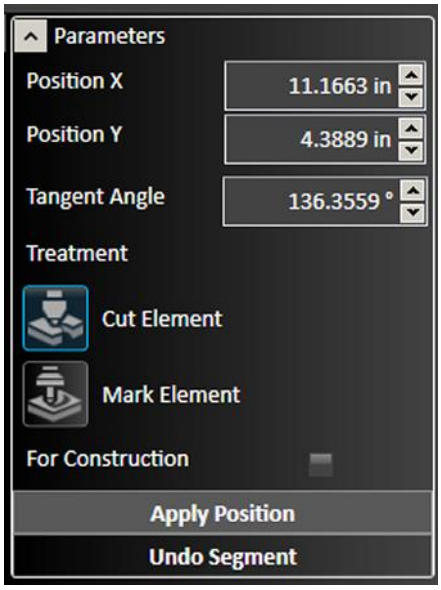
Specify the attributes of a specific line segment.

Apply Position

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

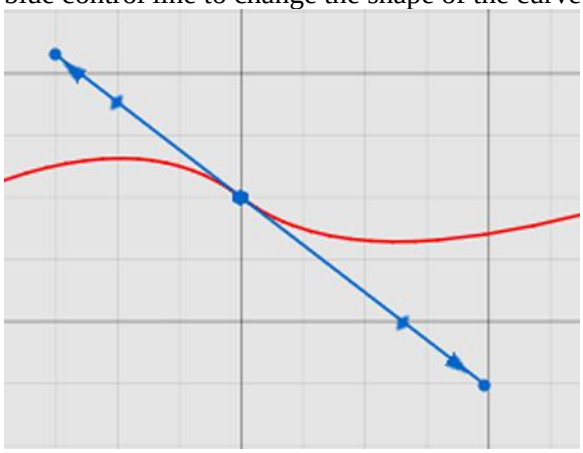
Undo Segment

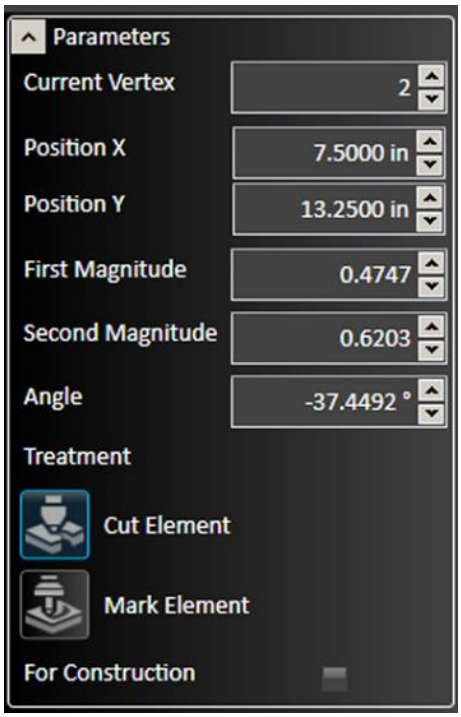
When clicked, removes the segment just added. Consecutive clicks remove additional existing segments.



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.





Current Vertex

Select the particular vertex on the spline.

Position X and Position Y

Specify the coordinates of the vertex.

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.

Angle

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

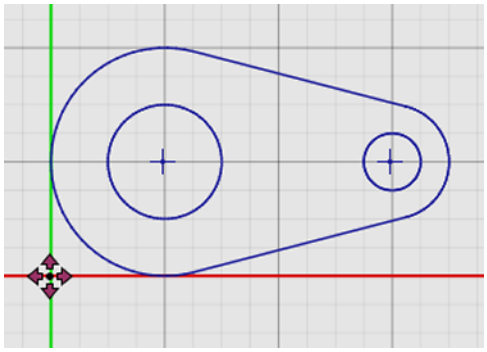
Shapes

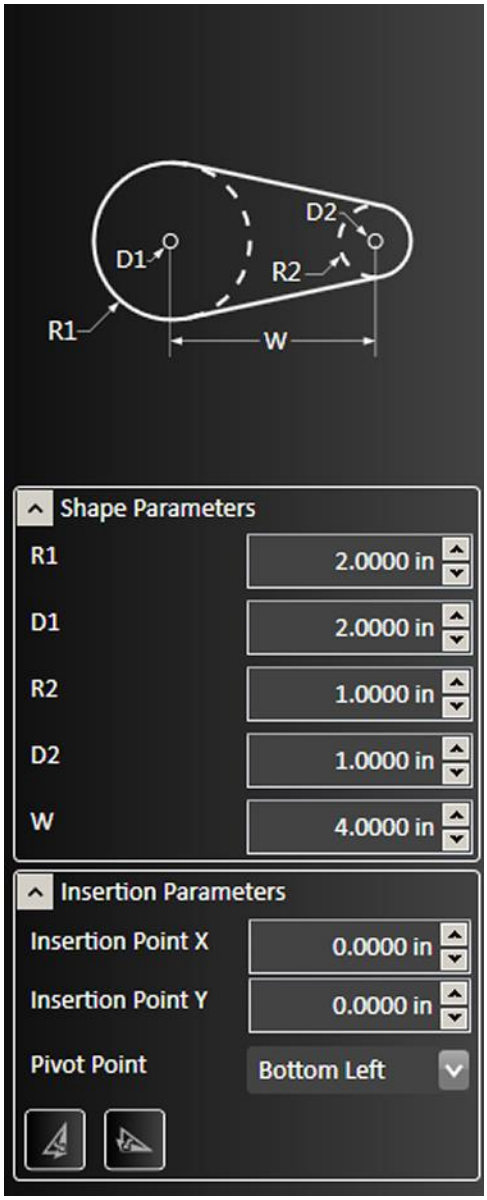
Shape Tool

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 the library in the parameters area.

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

Example: Two circles connected by tangents





Shape Parameters

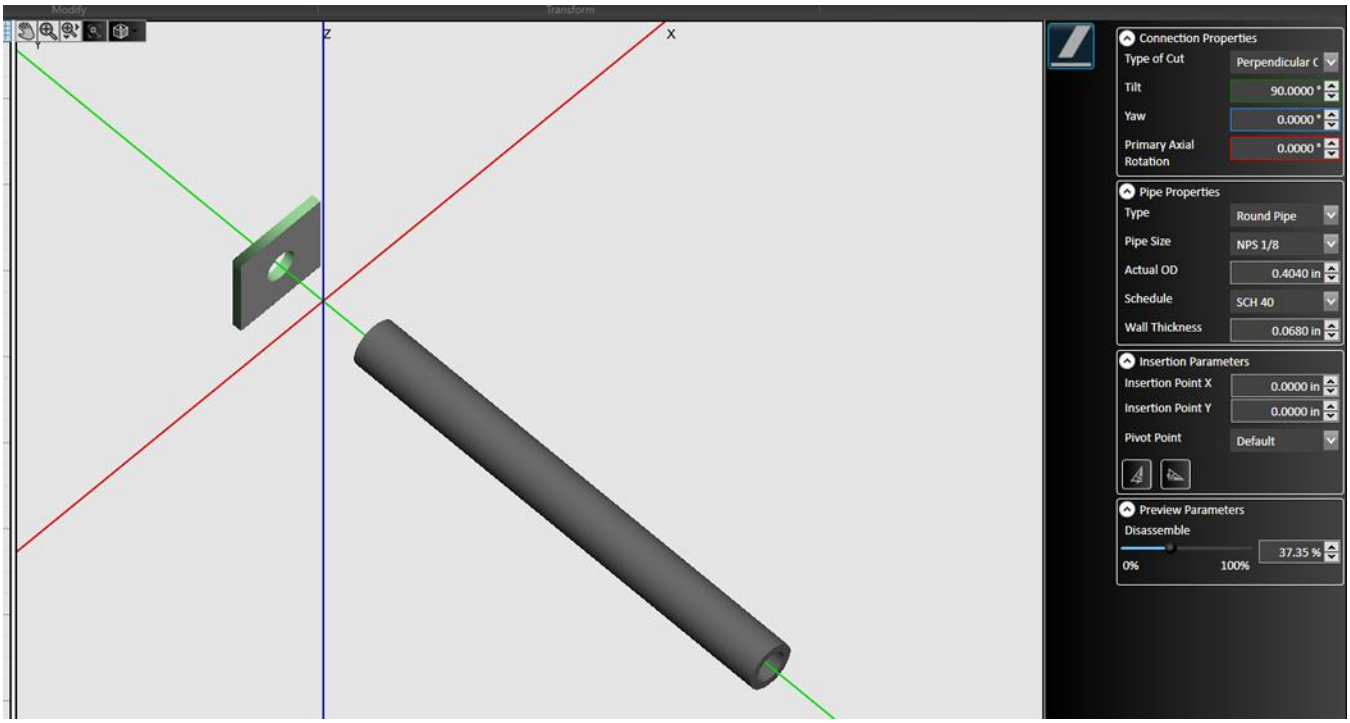
When the shape is imported into the drawing window, its specific parameters appear. For example, for the shape above, any of the following attributes can be manually adjusted: R1 and R2 (the radii of the two circles), D1 and D2 (the diameters of the two through holes), or W (the distance between the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

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.

Pipe Joint Shape Tool (2D part)

FlashCut CNC offers a set of preconfigured alternatives for visualizing and manipulating pipe joints, in drawings that contain pipe parts. For 2-D parts, the pipe joint shape tool shows the joint between the 2D surface and the pipe. The figure below shows the 3D preview window and the parameters pane.



Connection Properties

Configurable parameters include **Type of Cut, Tilt**, **Yaw**, and **Primary Axial Rotation**, in degrees.

Pipe Properties

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

Insertion Parameters

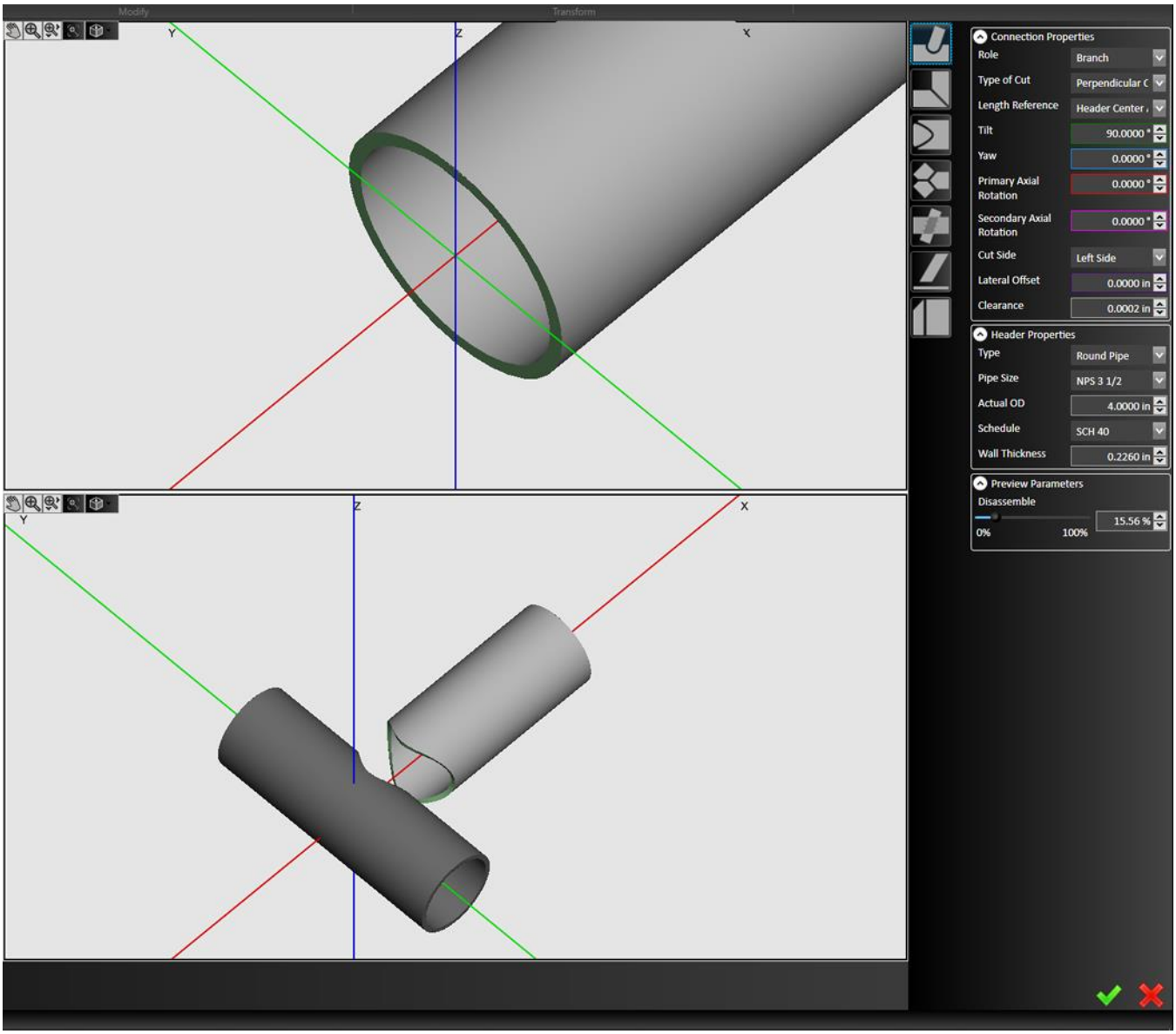
The position of the **X/Y Insertion Points** and the location of the **Pivot Point** can be specified. In addition, the shape can also be flipped horizontally or vertically.

Preview Parameters

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

Pipe Joint Shape Tool (pipe part)

For pipe/tube parts, the pipe joint shape tool provides alternatives for visualizing and cutting different types of pipe joints and cuts. Note that your machine axes and fabrication head must be correctly configured to create toolpaths for pipe/tube part drawings.

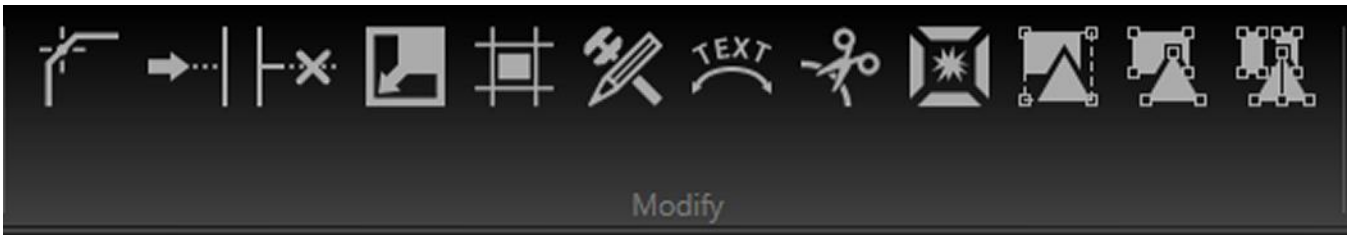


Joint Types

Available types are **Branch interaction** (shown in the figure above), **Miter interaction**, **End Cap interaction**, **Wye interaction**, **Pipe-through interaction**, **Pipe Branch interaction**, and **Straight Cope interaction**.

Modify

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

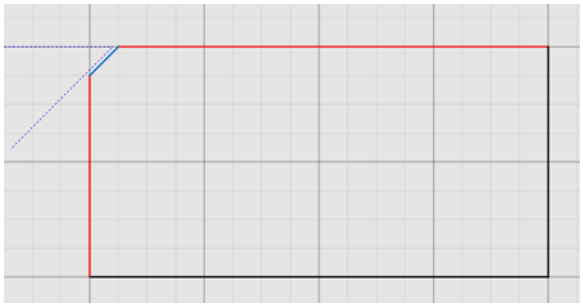


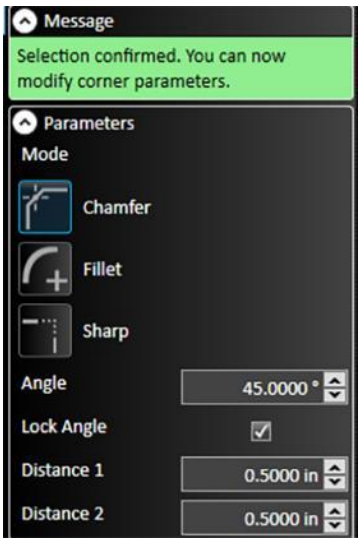
After modifications are complete, click the green check mark to **Apply** changes or the red X mark to **Cancel** changes.

Corner Tool

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

Apply Chamfer





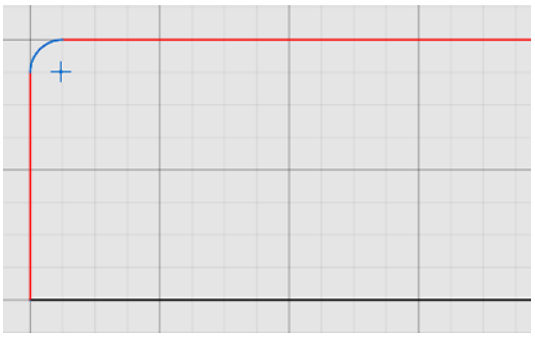
Select two intersecting lines or their intersection point, and then click **Chamfer** to add a sloped chamfer to this edge. You can scale the size of the chamfer by dragging the mouse towards or away from the chamfer, or in the parameter window, type the distances from the intersection in both directions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

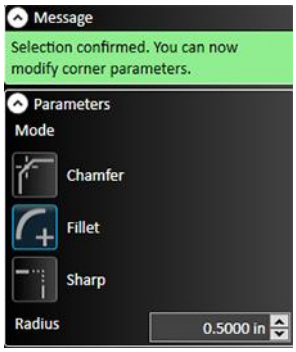
Choosing the **Lock Angle** option causes both distances to change when one of them is edited, holding the angle measurement constant. Once the chamfer is correctly configured choose the green check mark or press ↵**Enter**.

Selecting **Sharp** while editing will restore the corner to its original angle.

Apply Fillet

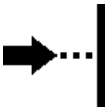
Select two intersecting lines or their intersection point, and then click **Fillet** to add a rounded fillet to this edge. You can scale the size of the fillet by dragging the mouse towards or away from the fillet, or in the parameter window, type in the radius of the fillet. Once the fillet is correctly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



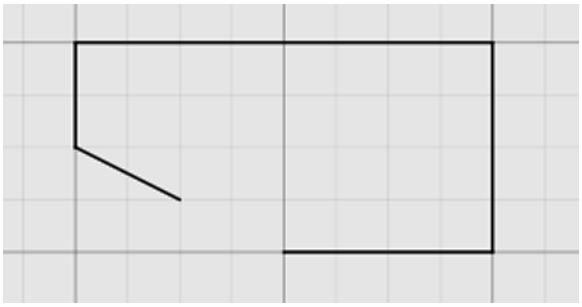


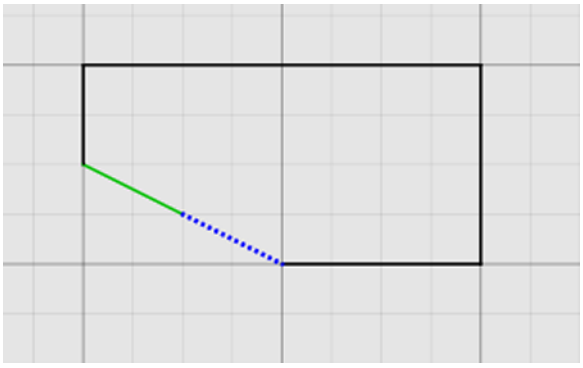
Selecting **Sharp** while editing will restore the corner to its original angle.

Extend Tool



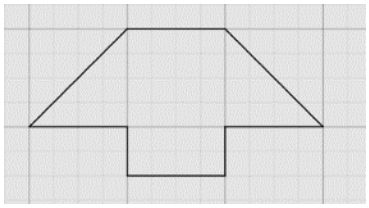
Click the **Extend Tool**, and then hover the selection cursor over the endpoint of the line or curve to be extended. A preview of the extension should appear (in blue, below) to show where the line or curve will intersect with another object in the drawing. Click the mouse to finalize the extension.

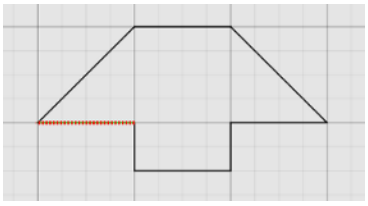




Trim Tool

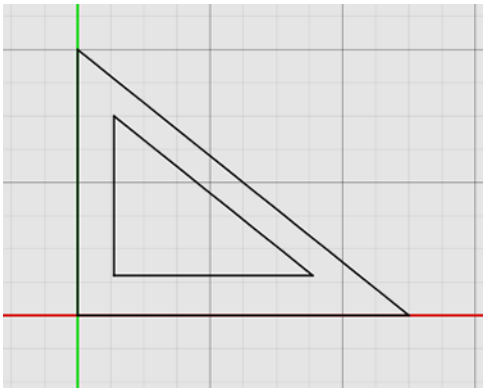
Click the **Trim Tool**, and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear (in red, below) where the line or curve will be removed. Click the mouse to finalize the trim.

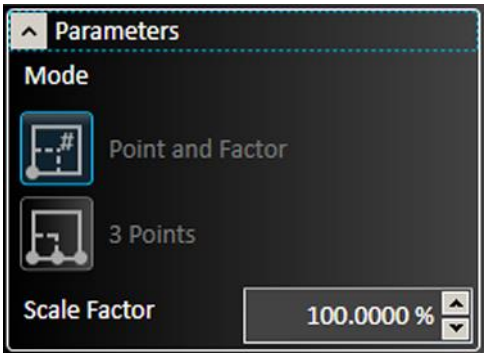


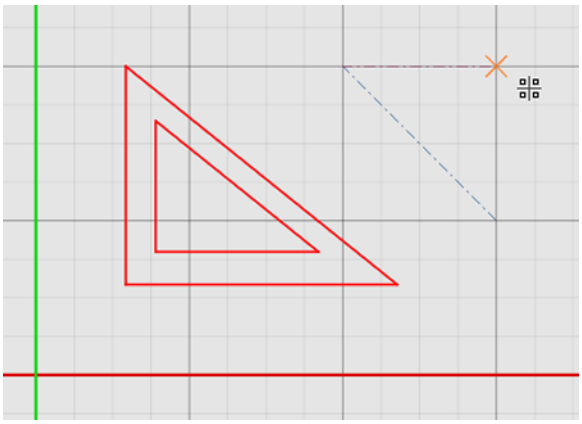


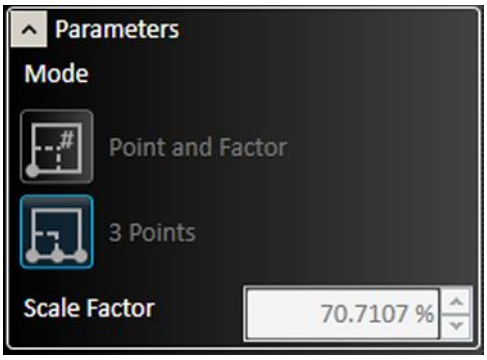
Scale Entities

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

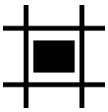






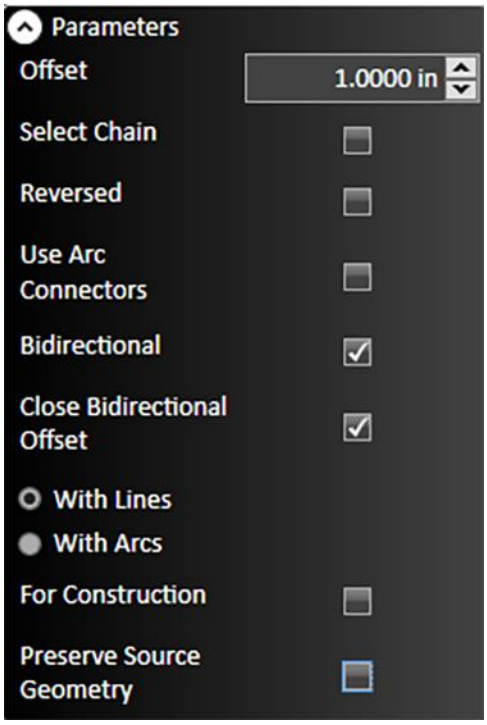


Apply Offset



Select that feature and then click **Apply Offset**. You may then select parameters for the offset in the parameter window.

The offset tool duplicates selected features and positions them at the specified distance away from the original feature.



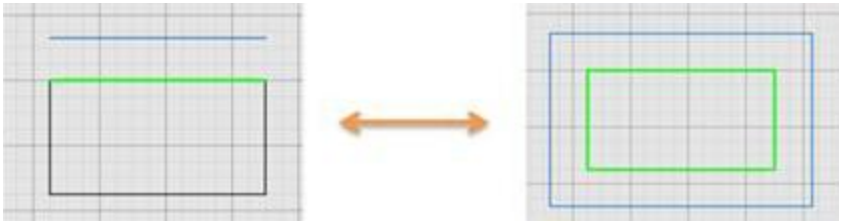
Offset

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

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**

Reversed

Reversed changes the direction of the offset distance.

Use Arc Connectors

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

Bidirectional

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

Close Bidirectional Offset

Close Bidirectional Offset connects the offset features using either lines or arcs.

For Construction

Select to make the offset feature a construction line.

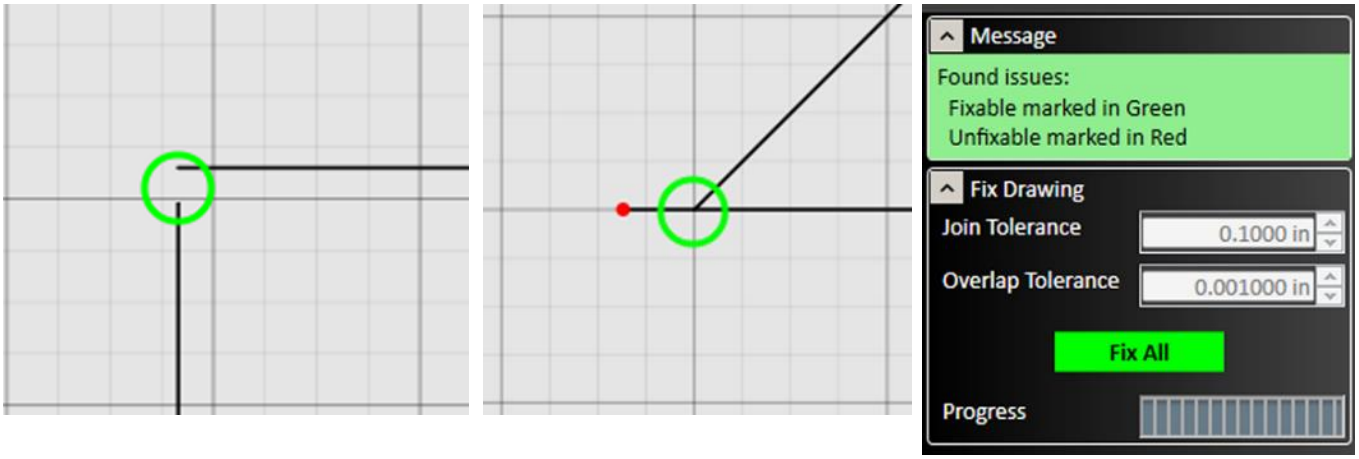
Preserve Source Geometry

Prevents the original feature from being altered.

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.

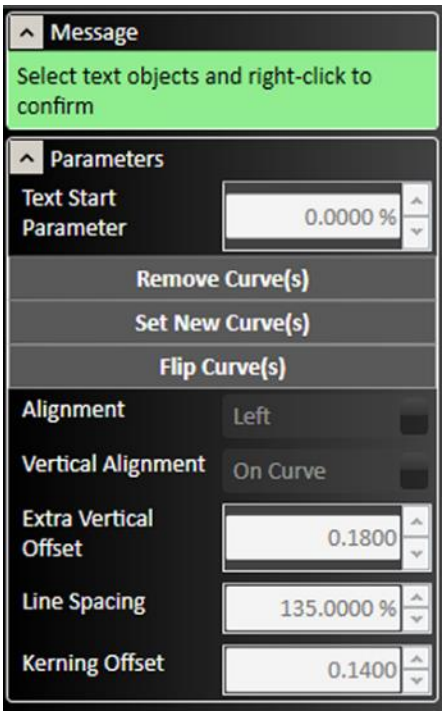


Use the **Search Issues** button to identify issues with the drawing that could lead to mistakes in fabrication. Issues noted with a green circle can be fixed automatically by selecting **Fix All**. Issues noted with a red circle must be addressed manually.

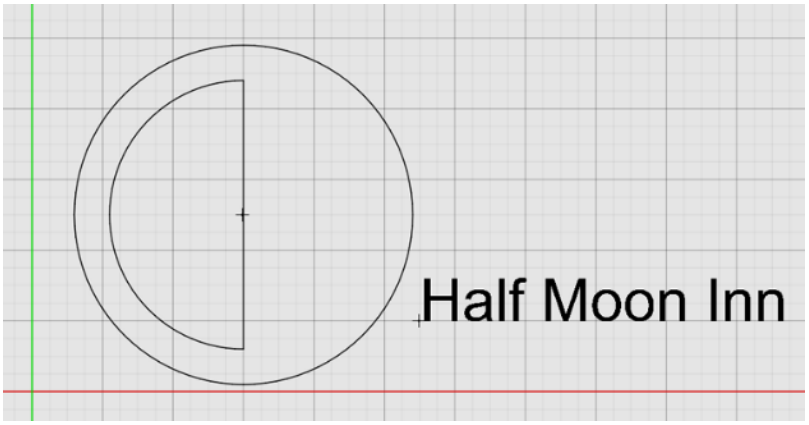
Curve Text

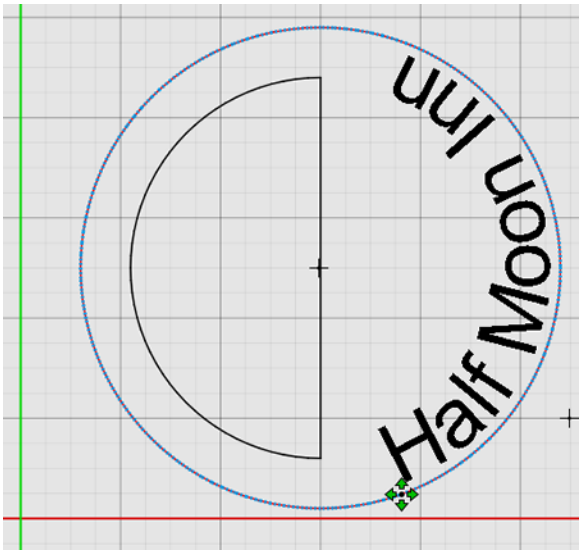
The **Curve Text** tool modifies an existing text object to follow a curved shape. Start by creating both the text object and the curved shape you want to use.

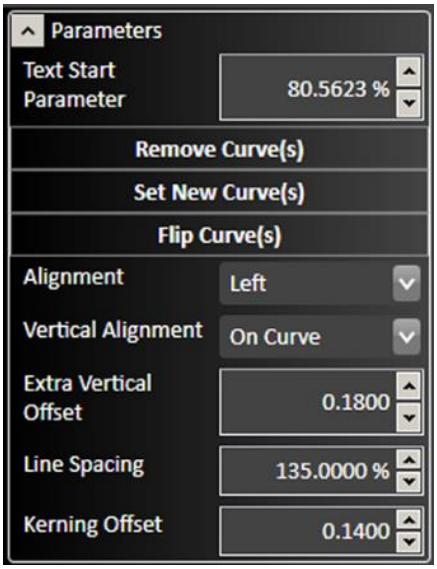
Select both the text object and the curve object in the viewport, then click **Curve Text**. The text will be automatically aligned with the curve. If the text object and curve object are not already selected, the tool will provide instructions for how to select the text object and curve object. The position of the text can be adjusted with the cursor and modified by changing the alignment parameter.



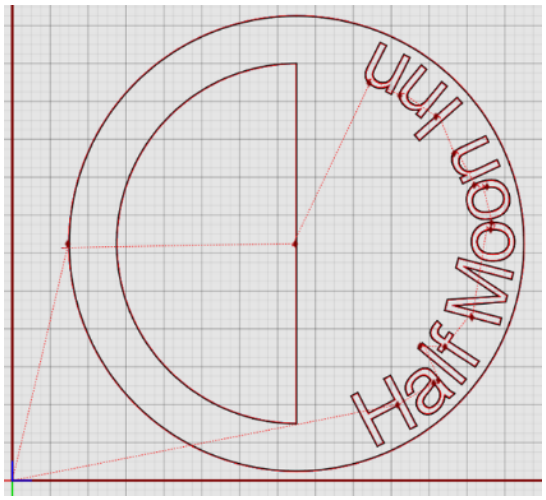
In the example below, note that the **Extra Vertical Offset** and **Kerning Offset** have both been adjusted to make the curved text even and legible.







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)

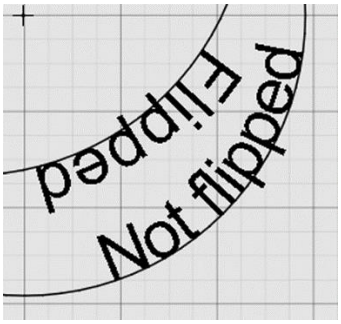
Separates the text object from the curve.

Set New Curve(s)

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

Flip Curve(s)

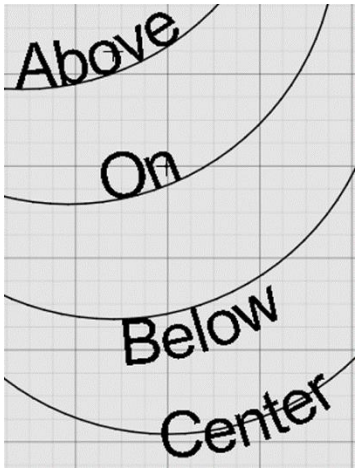
Changes the side of the curve that the text object follows.



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 is always 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 object and the curve.

Line Spacing

Affects the spacing of multiple lines of text.

Kerning Offset

Adjusts the spacing between letters.

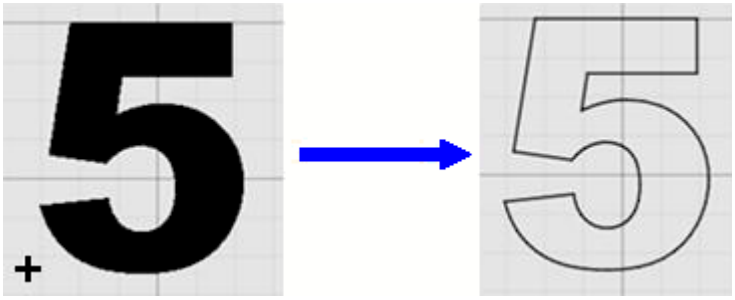
Cut Tool

Use the **Cut Tool** to divide an entity such as an arc or a line segment into multiple sections. Select the **Cut Tool** and click the mouse at the position on the entity where you want the cut to be made. You can verify the cut by hovering the cursor over the feature. Only a part of it will highlight if the cut worked properly.

Explode Entities

Transforms a feature that is solid and not composed of lines (such as a text object) into individual line segments. This is necessary for toolpath generation of any feature that is solid.

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



Group

The **Group** command combines selected objects so that they can be modified or transformed as a single unit.

If you choose this command with no objects selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking.

Ungroup

The **Ungroup** command separates the selected grouped objects.

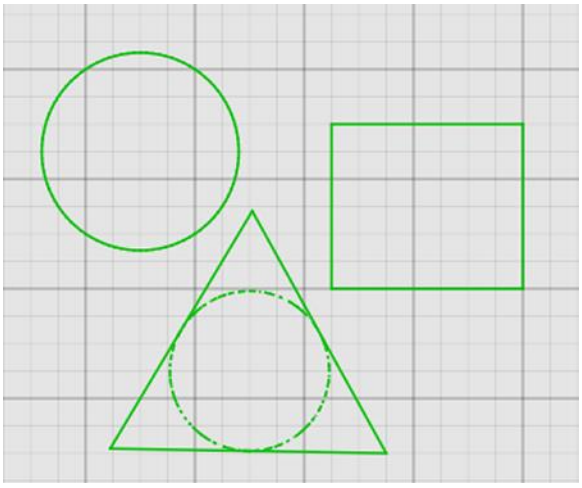
If you choose this command without a group selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking. See

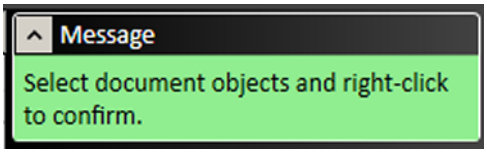
Ungroup All

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.

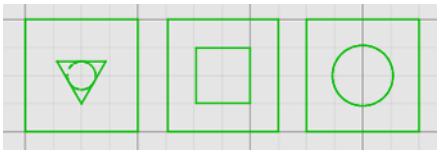




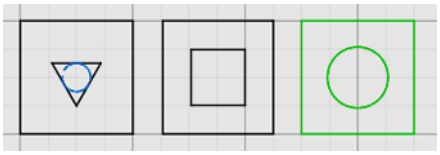
Ungroup vs. Ungroup All

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.

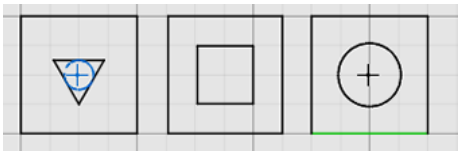
Each object in the group is itself a group of multiple objects.



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



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



Transform

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 **Apply** 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

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

Delete Selected Objects

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

Move Entities

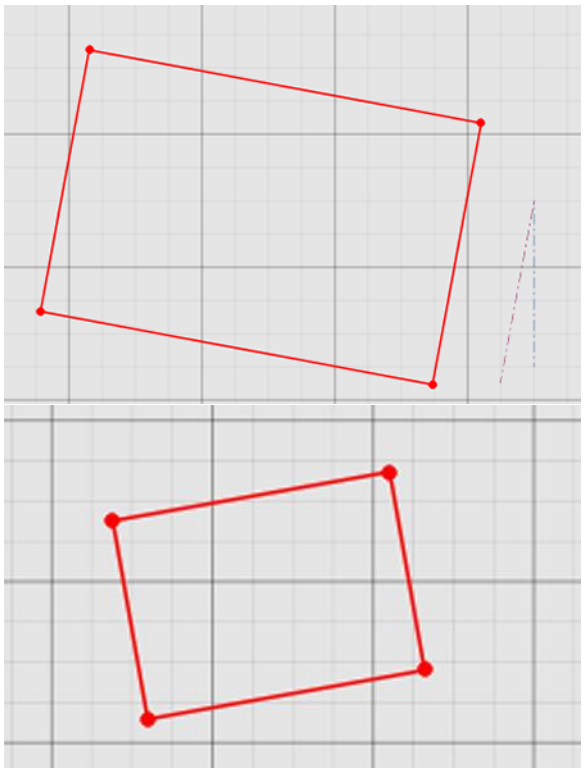
Select the features to be moved, and then click the **Move Entities** tool. Click the mouse to set a reference point, and then move the cursor to move the feature(s) in relation to that point. Click the mouse again to select the new location for the features.

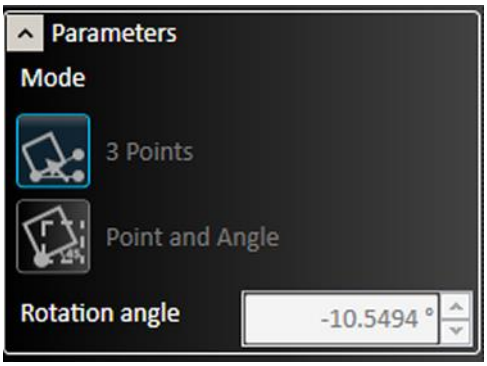
Rotate Entities

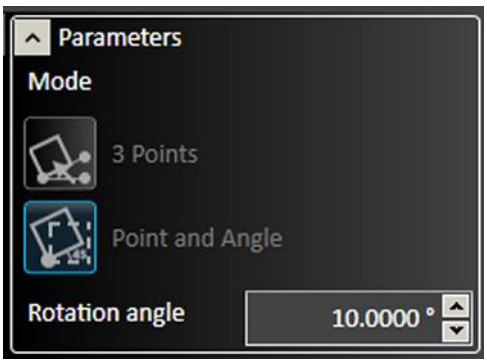
FlashCut CNC offers two modes for rotating features in relation to a set point.

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

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

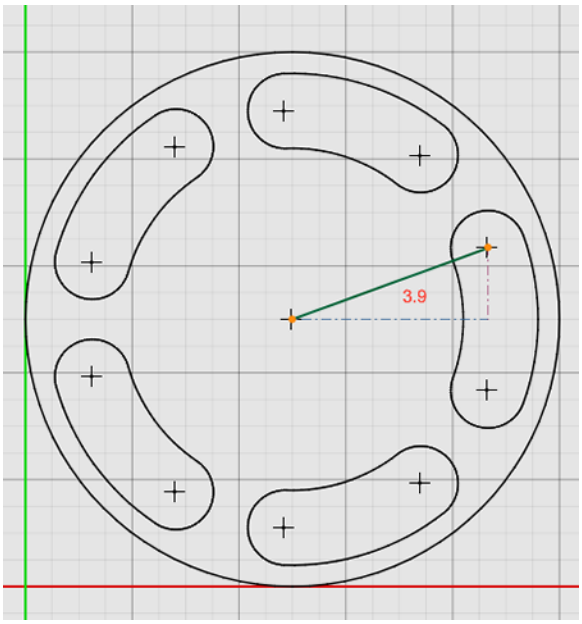


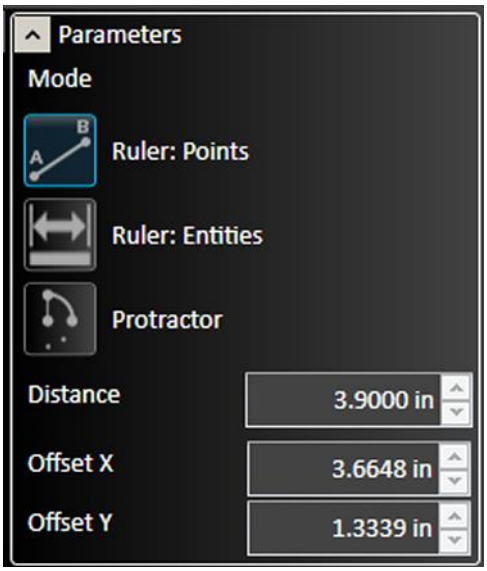




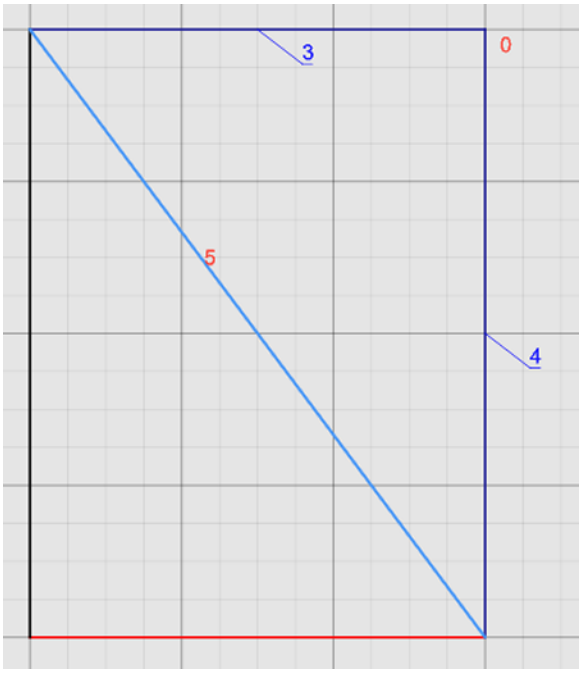
Measure Entities with Ruler

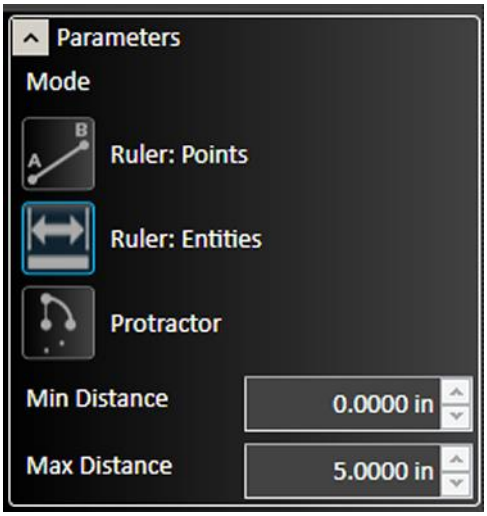
You can measure entities in one of three modes: as a **ruler** between **points** or between **entities**, or as a **protractor** between points on an arc. The tool allows for measurement between any two points and displays offset in X and in Y directions.



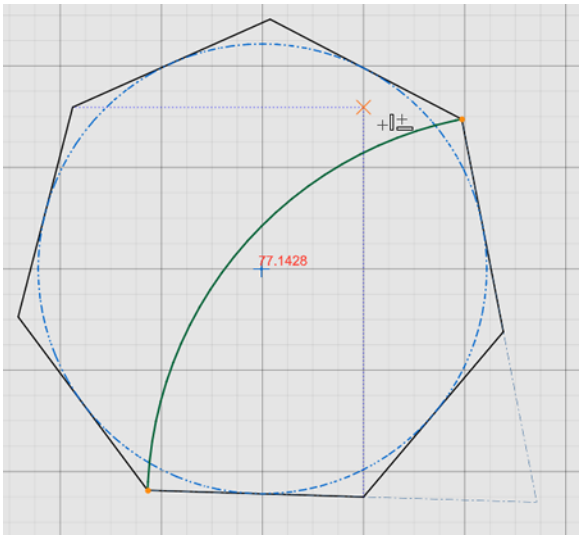


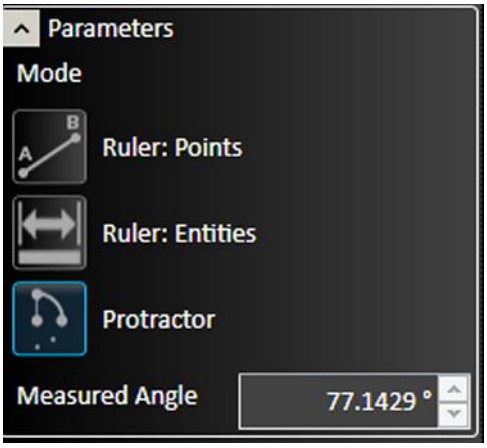
Ruler used to measure between the distance between points: center of an arc, and radius of an arc. Note the construction lines marking the other sides of the triangle: these represent the **Offset X** and **Offset Y**





Ruler used to measure the distance between two 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 sides.



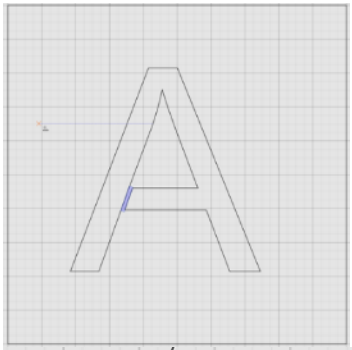


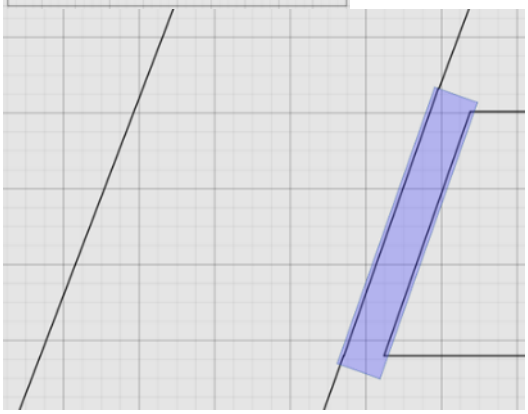
Protractor used to measure the arc and **Measured Angle** between two points of a polygon. Note that the protractor will not measure an arc or angle if it is grouped with other lines or shapes. You must ungroup the arc to enable the protractor tool.

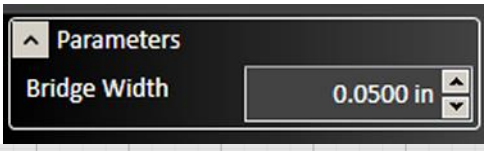
Bridge Entities

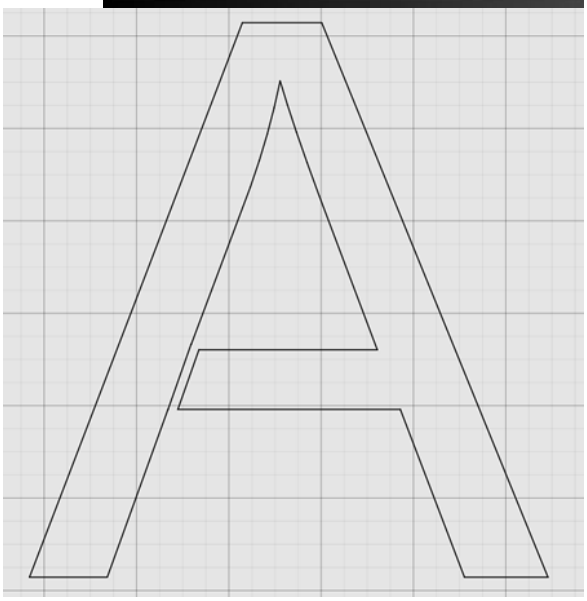
Connects two or more objects with a bridge that spans the gap between them. Can also be used to divide a single object into separate objects. The width of each bridge can be set individually.

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





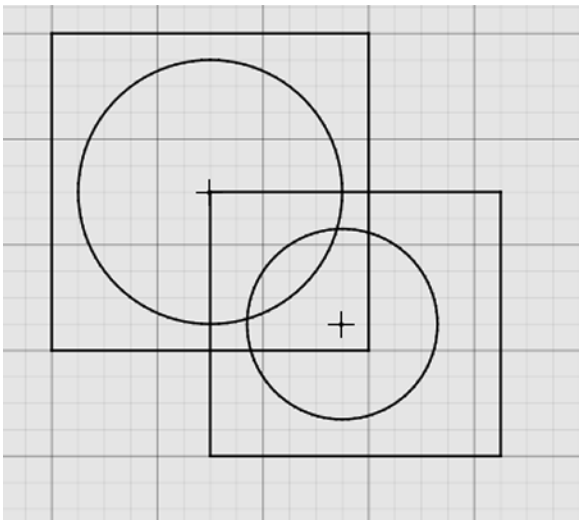


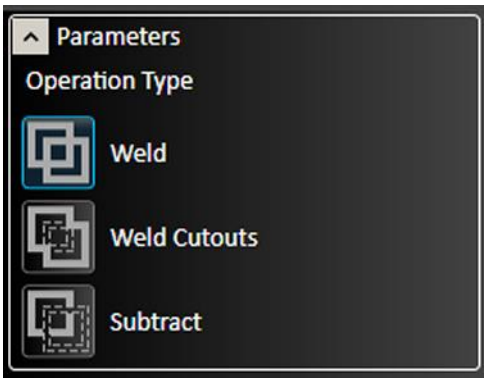


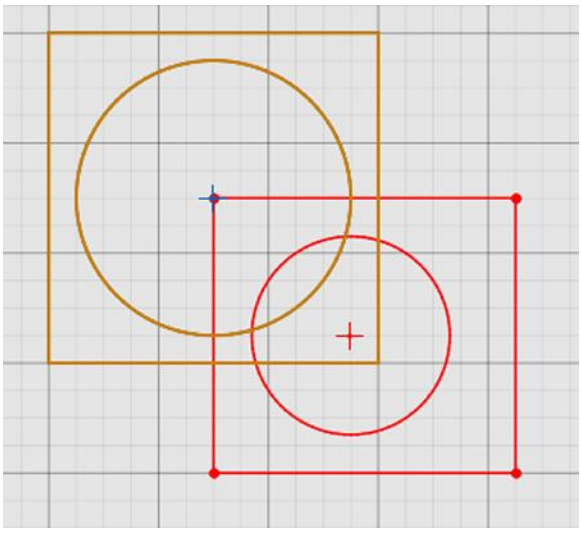
Perform Boolean Operation on Entities

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

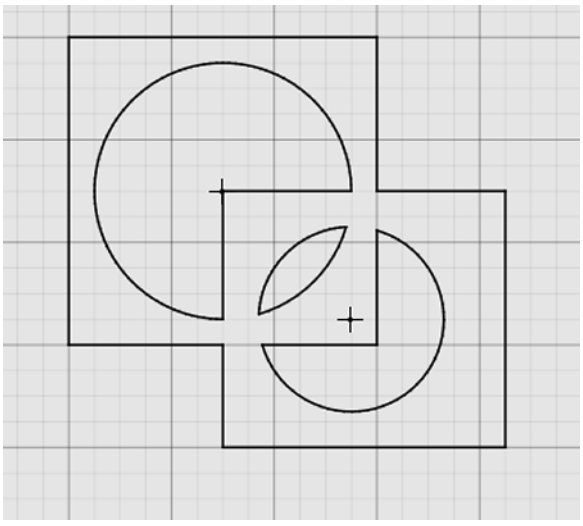
Select Perform Boolean Operation, select the specific operation in the parameters window, then select the objects one at a time, right clicking to confirm each selection. The first object selected will be modified by the second. The example below shows the results of all three operations on the initial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



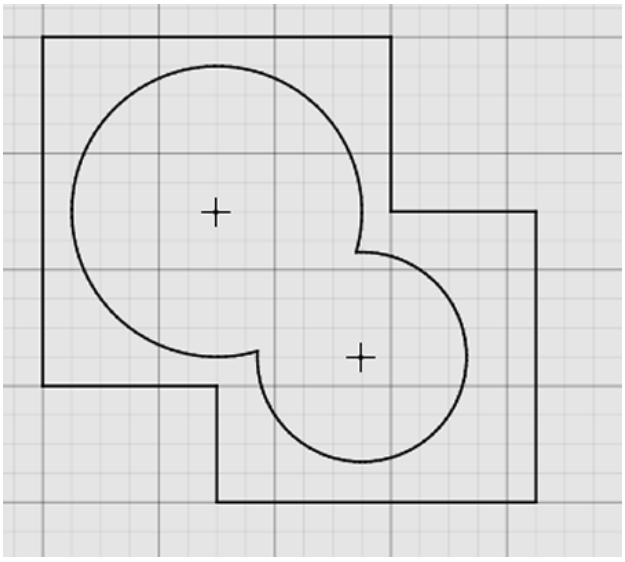




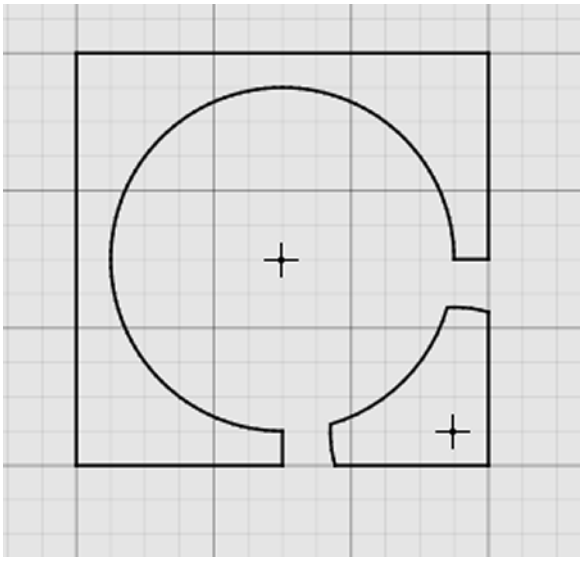
Shapes highlighted to indicate which object is first (gold) and second (red).



The results of a **Weld** operation.



The results of a **Weld Cutouts** operasion

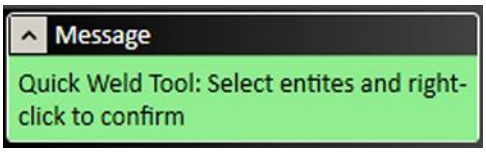


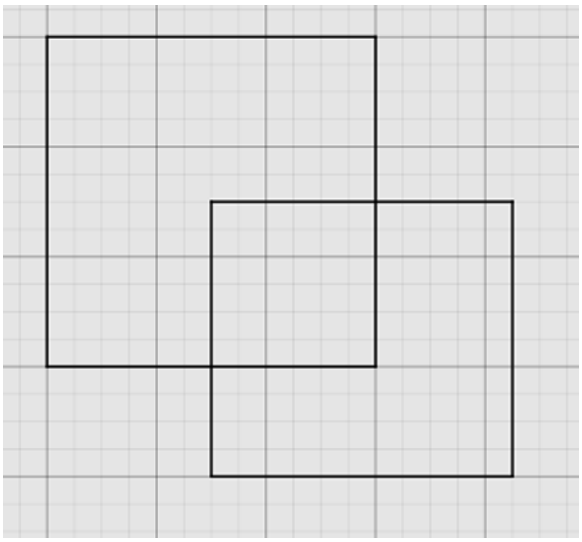
The results of a **Subtract** operation.

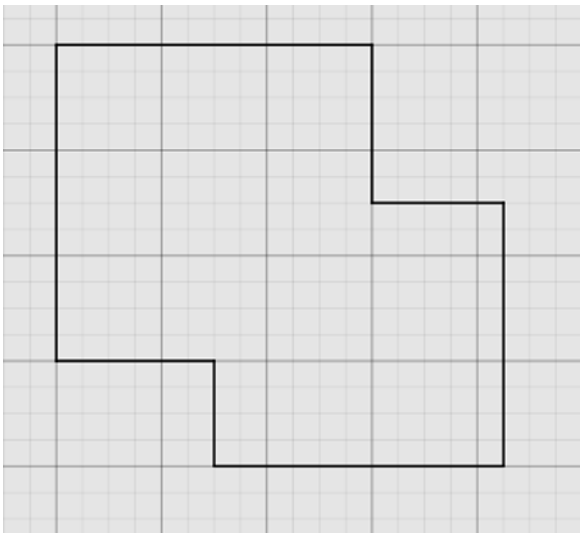
Weld Selected Entities

To quickly combine multiple shapes into a single shape, first create the desired overlapping objects.

Select the **Weld Selected Entities** tool, select the objects, and right-click to confirm the operation.





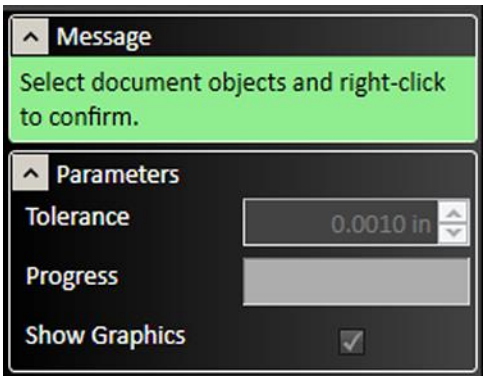


Arc Fitting Tool

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

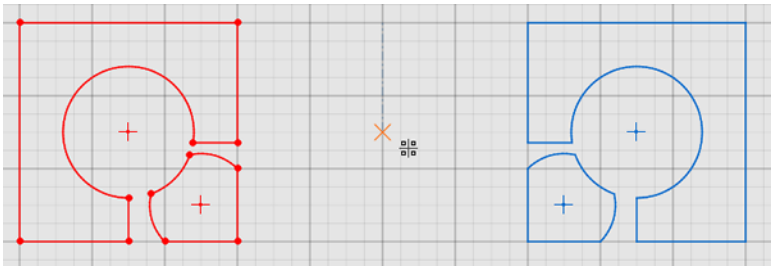
Tolerance can be set in drawing units. A lower number will increase accuracy. **Progress** during recalculation is shown in the bar.

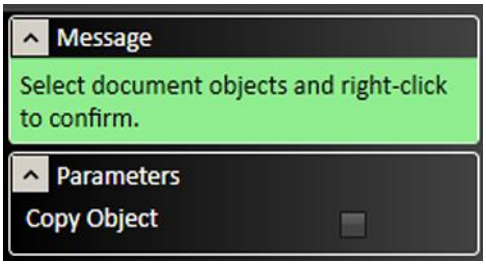
When **Show Graphics** is selected, an outline lines and arcs will be displayed in the drawing window.



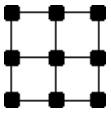
Mirror Entities

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





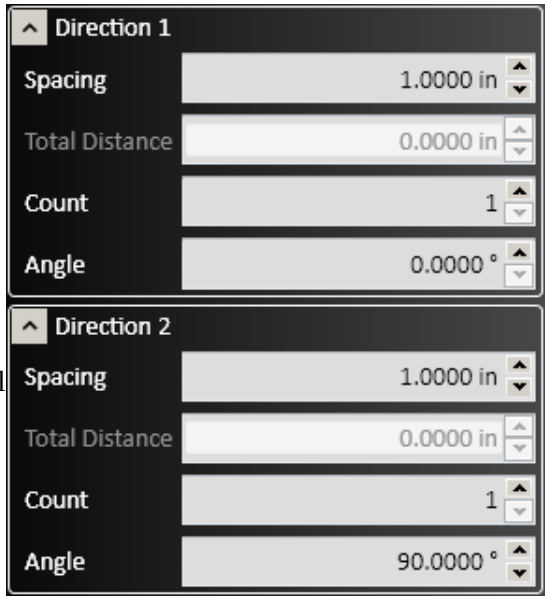
Linear Pattern Tool

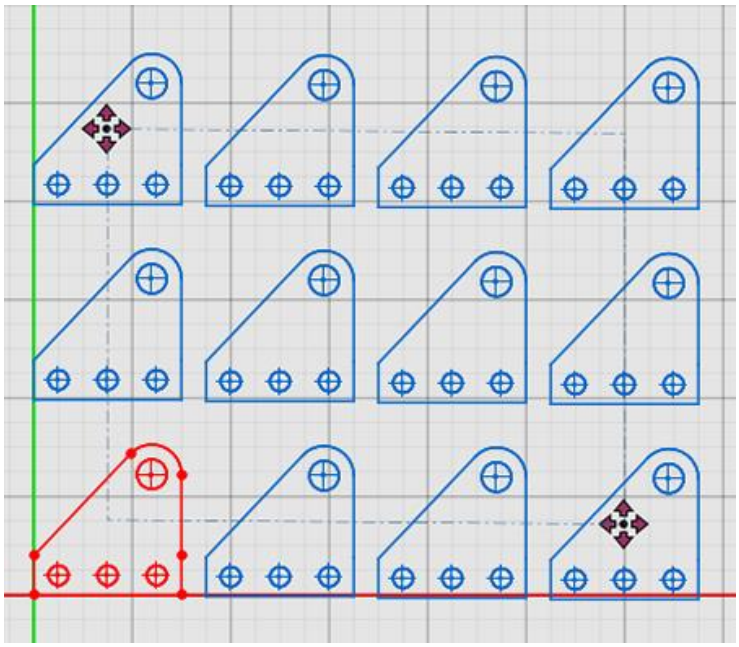


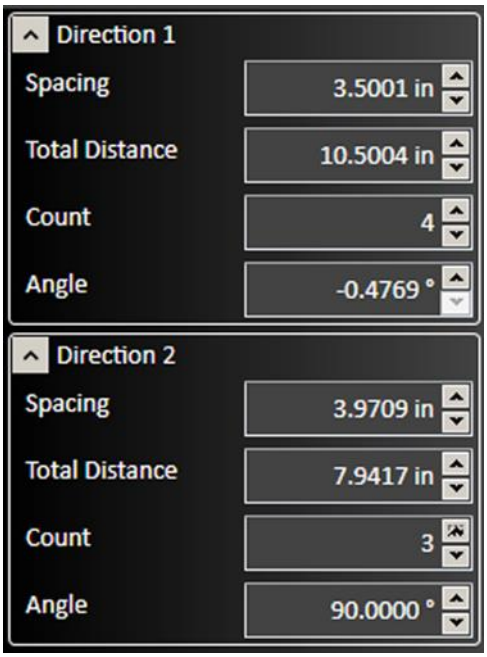
The linear pattern tool duplicates selected features in a grid fashion. Select the feature to be duplicated, select the **Linear pattern** tool then specify parameters.

Alternately, click and drag the blue points on the drawing screen to select the parameter for **Total Distance**. After changes are complete, click the green check mark or press ↵**Enter** to accept changes or the red X mark to cancel changes.

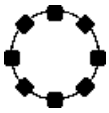
- **Direction 1** represents the horizontal direction.
- **Direction 2** represents the vertical direction.
- **Spacing** sets the distance between copies.
- **Total Distance** represents the overall distance from the original to the final copy.
- **Count** indicates the number of copies to generate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



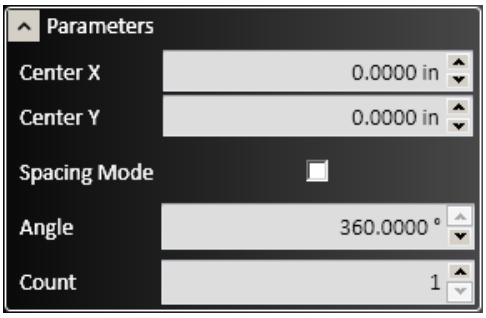




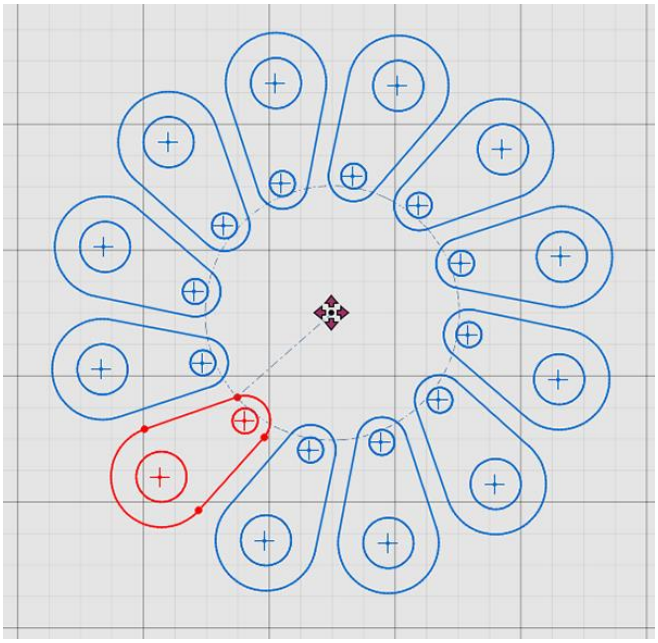
Circular Pattern Tool

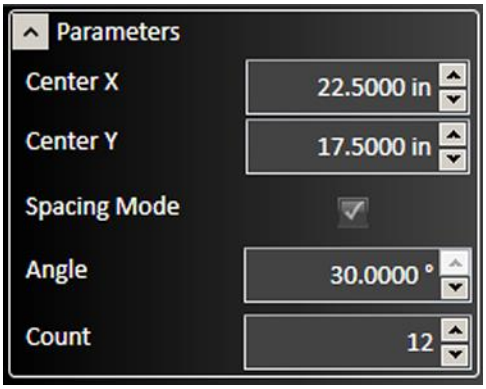


The circular pattern tool duplicates selected features in a circle around a specified point. Select the feature to be duplicated, select the **Circular Pattern** tool, and then specify the parameters.



- 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.





Manipulation Tools

Any objects or set of objects in FlashCut CAD can be stretched, positioned, or rotated by manipulating a set of control points with the cursor, or by entering values using the toolbar.

Note that different types of objects respond differently to scaling. For example, circles will expand, and straight line segments will be extended, but ellipses and arcs will be divided into multiple line segments.

Control handle grid

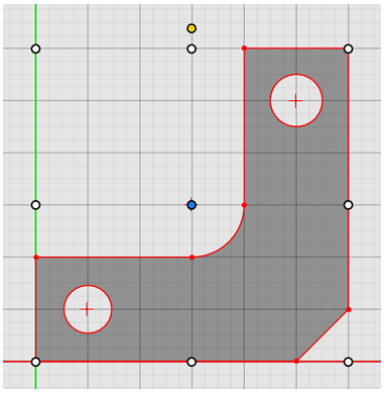
Selecting an object or objects in the drawing workspace displays a grid of control points.

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 point.

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

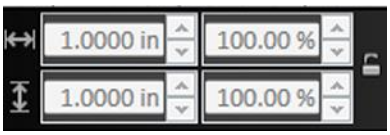


Horizontal and vertical dimensions

The horizontal and vertical dimensions of the area under control are shown in two 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.

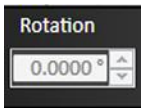
Click the lock icon to lock the aspect ratio.



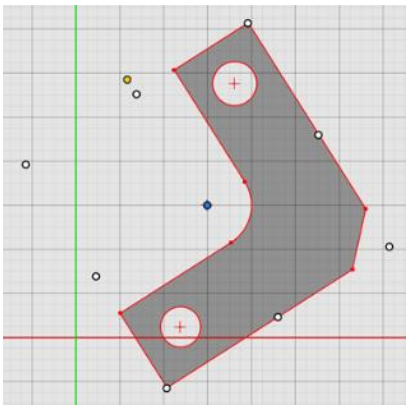
Rotation

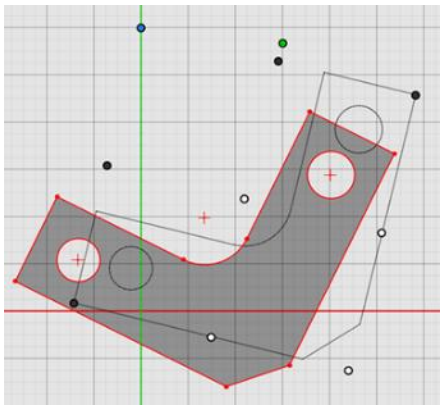
The selected (blue) control point defines the center of rotation.

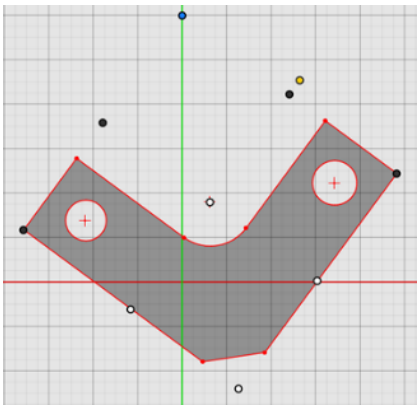
The degree of rotation is displayed in the field. Changes can also be made directly to the field.



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.







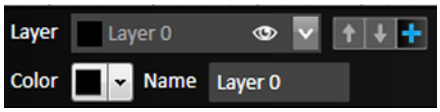
X and Y position

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

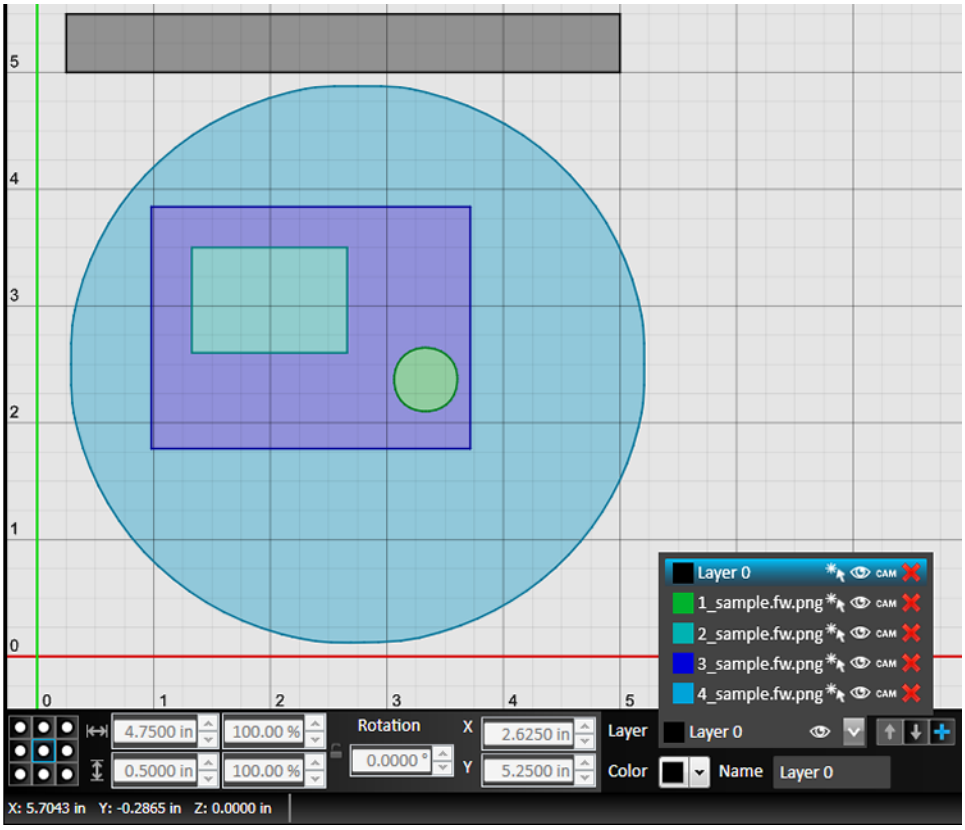
Changes can also be made directly to each field

Layers

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



An example of the use of Layers is shown below where 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.

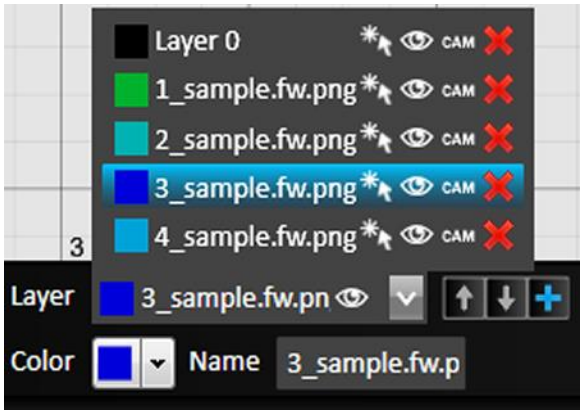


Layer Controls

The information about the active layer is always displayed in the toolbar. When the layer menu is expanded, additional details are shown, and additional controls are available, for each layer. Note how the active layer is highlighted.

From the expanded layer menu, you can:

- View the Color and Name of each layer in the drawing.
- Select all objects in that layer.
- Show or hide all objects in that layer.
- Enable or disable the use of this layer in CAM. If disabled, CAM will not generate the toolpath for that layer.
- Remove that layer.



The Move Up and Move Down arrow buttons can be used to move layers up or down in the stack.

The "+" Add Layer button will add a new Layer.

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.

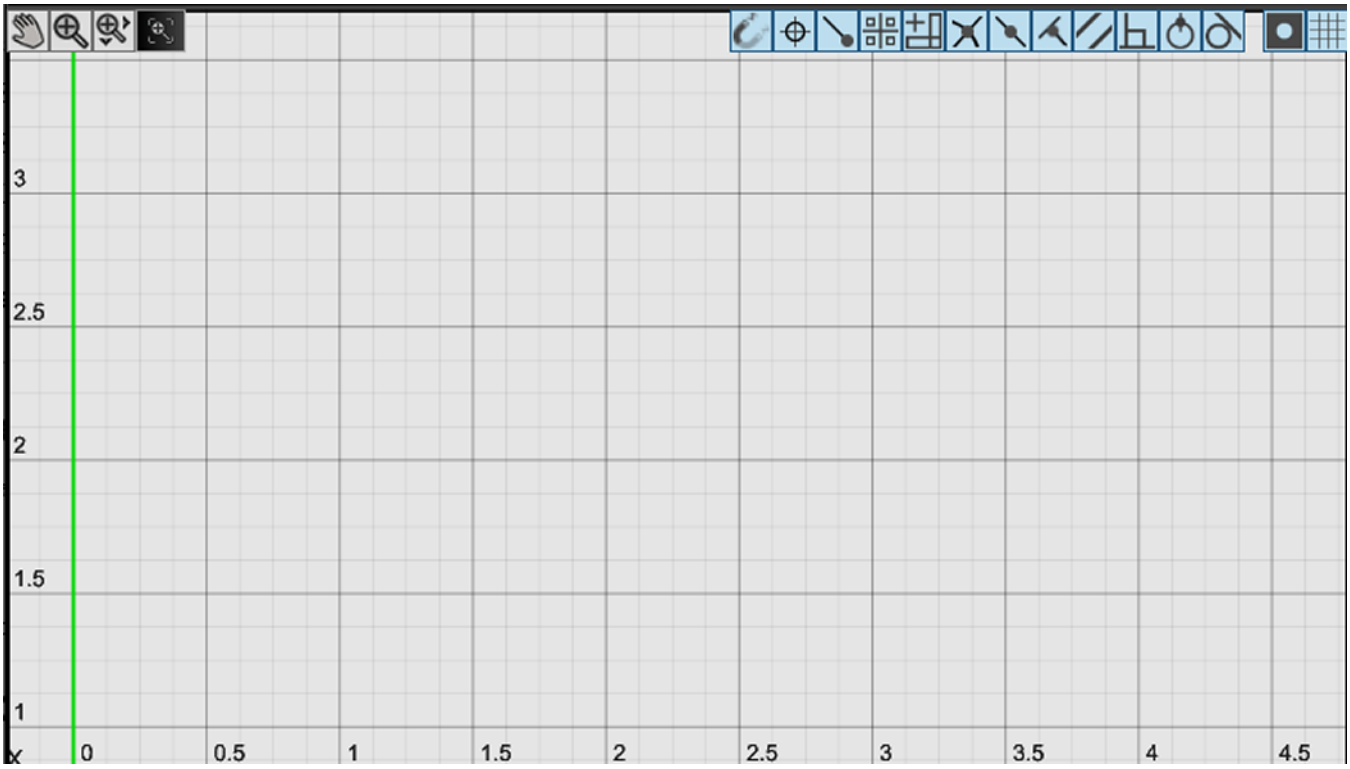
The **Name** of the active layer can be edited here.

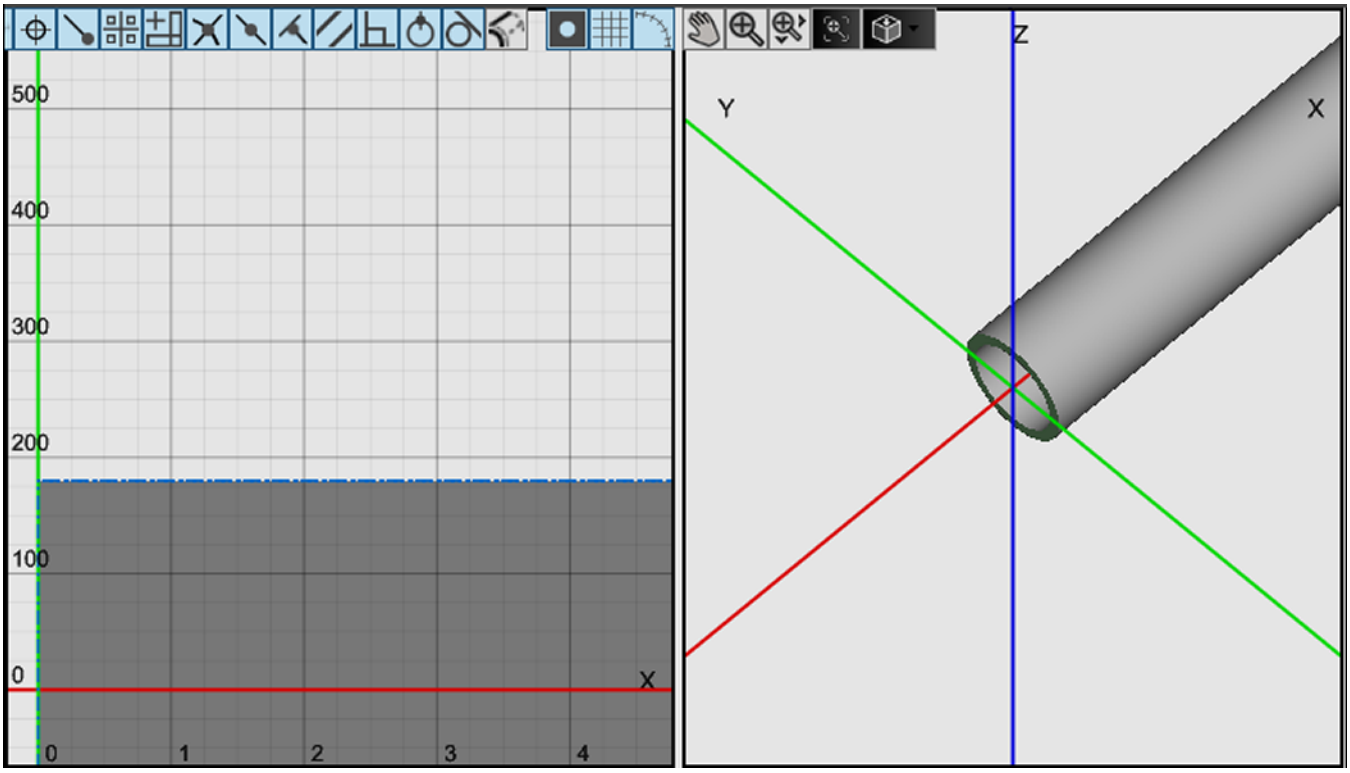
Combining Layers

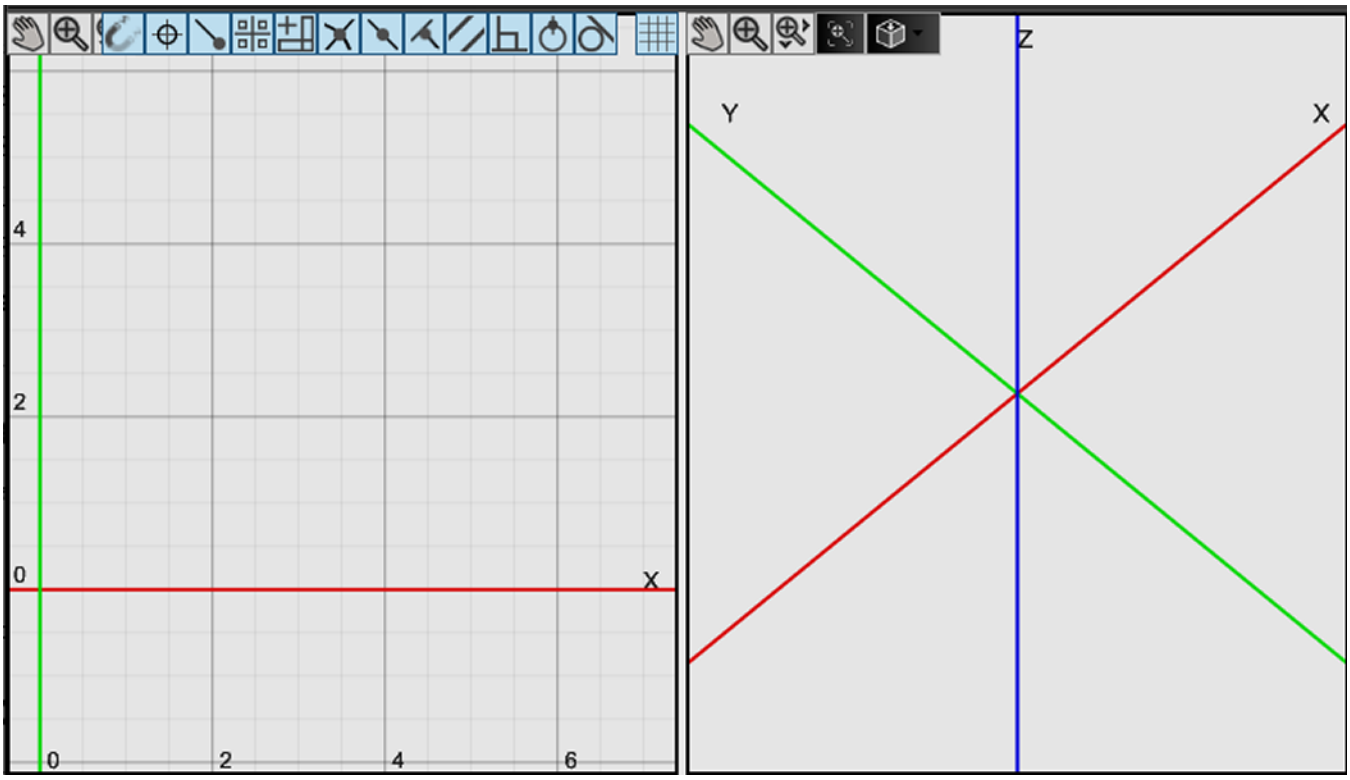
If you would like to combine layers, simply select all objects that you would like to have on a single layer. Notice that when you select multiple objects that are on different layers, the Layer menu will be blank. Expand the Layer menu and select the desired Layer that you want those objects to be on.

Viewport for CAD

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.







Pan and Zoom Tools

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

Pan

Click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window allows you to pan without selecting the pan tool first.

Zoom

Click and drag anywhere to zoom in and out on the drawing. You can also use the scroll wheel to zoom; scroll up to zoom in and scroll down to move out.

Zoom Window

Click and drag a box in which to zoom in. This tool allows a user to focus on a specific area.

Zoom Extents

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

3D Orientation Tool

The right-hand viewport 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.

Snap Tools

The snap tools affect how drawing tools interact with the background grid or with features already created. Clicking a snap tool toggles it on or off. Snap tools cause a feature or segment to lock on to the closest active snap location when the cursor is dragged nearby. Note that more than one type of snap can be active at one time.

Note that the snap tools can also be reviewed and selected by right clicking in the drawing window. This brings up a contextual menu, including a sub-menu to control snaps.

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

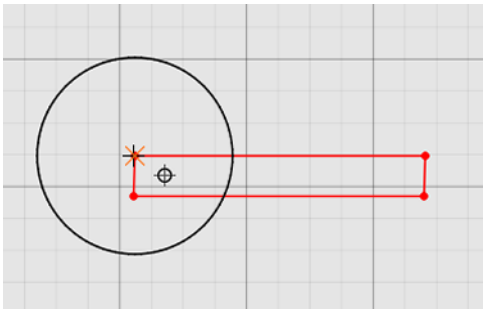
All Snap

When All snap is selected, individual snap options may be selected.

No snap can be selected when All snap has been deselected.

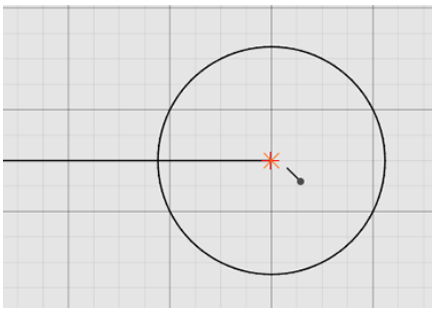
Center Snap

Features connect to the centers of elements.

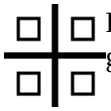


Endpoint Snap

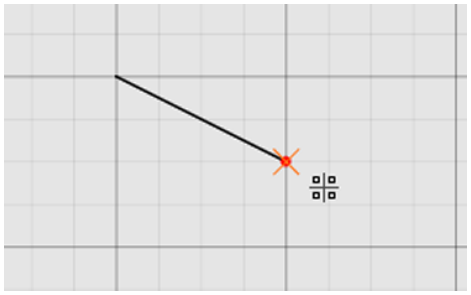
Features connect to the endpoints of elements.



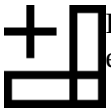
Grid Snap



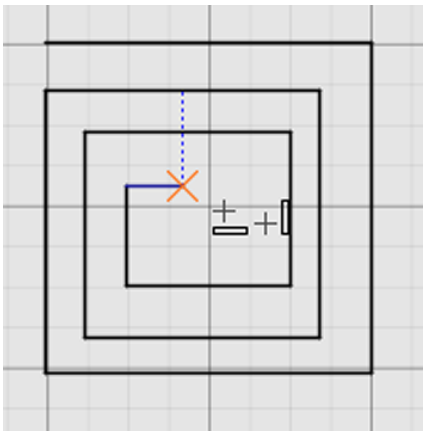
Features connect to points located along the background gridlines.



Horizontal/Vertical Snap

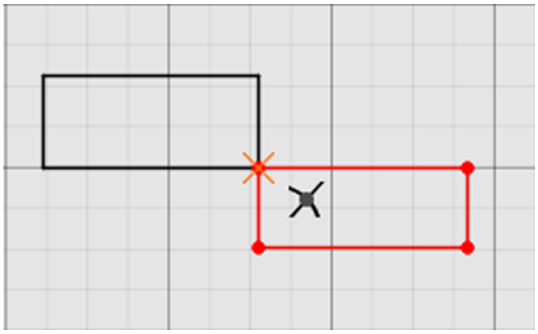


Features align vertically and horizontally when pulled into either direction.

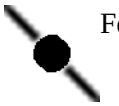


Intersection Snap

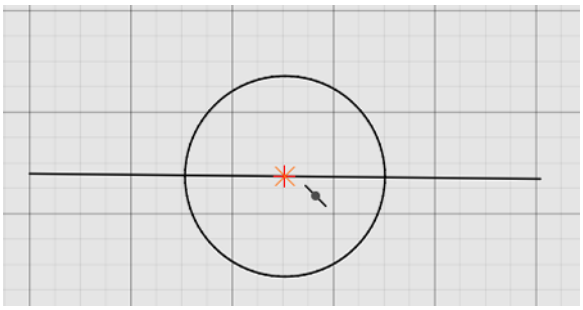
Features connect to the intersection points of elements in the drawing.



Midpoint Snap

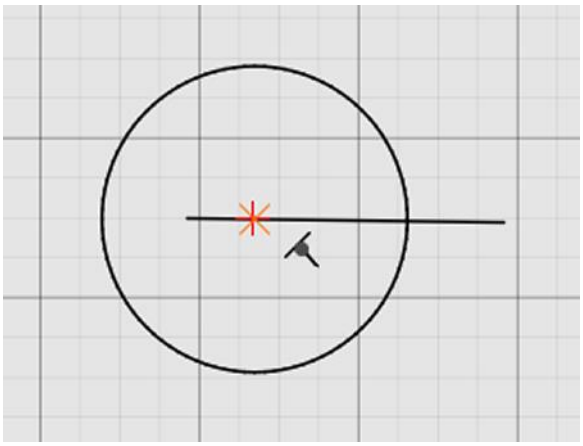


Features connect to the midpoints of elements.



Nearest Snap

Features connect to the element closest to the cursor.

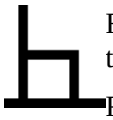


Parallel Snap

Features snap to a line parallel to the selected element.

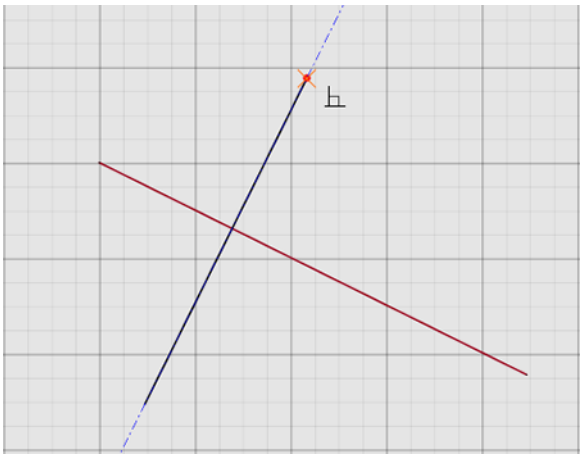
Right: a blue dashed construction line indicates that the bottom is parallel to the red highlighted line.

Perpendicular Snap



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

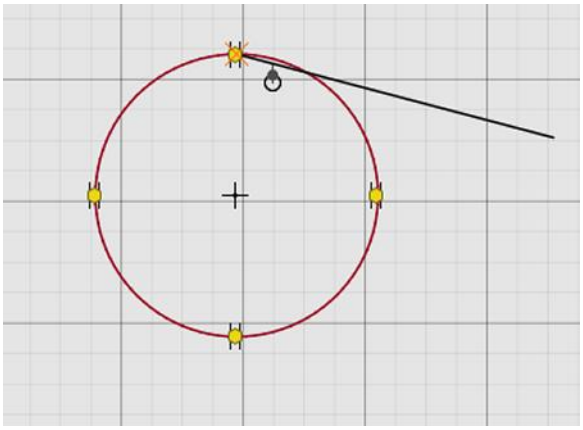
Right: a blue dashed line indicates that the line is perpendicular to the red highlighted line in the drawing.



Quadrant Snap

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

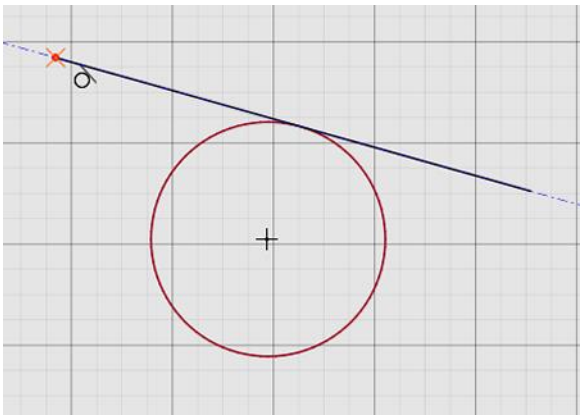
Right: line connected to a point at the top of the circle.



Tangent Snap

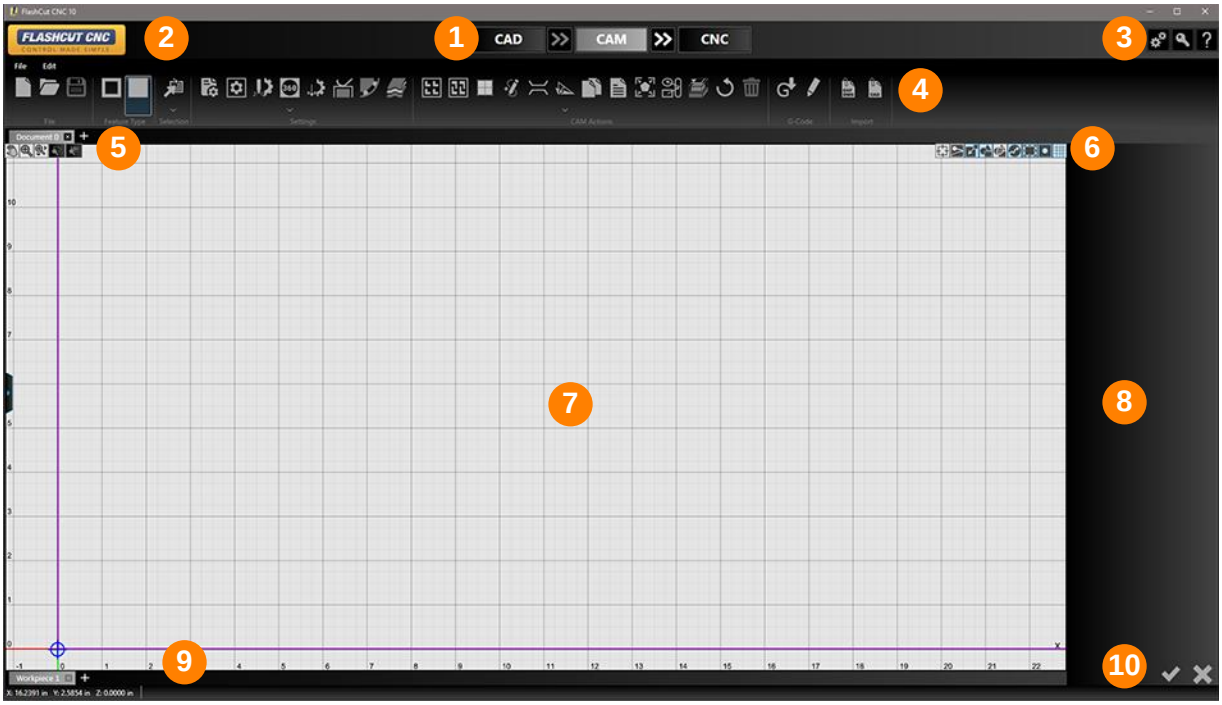
Features snap to a line tangential to elements in the drawing.

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



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.



1. Navigating Through CAD CAM CNC
2. Menu Bar
3. Configuration License Help
4. CAM Ribbon
5. Document Tabs
   Pan and Zoom Tools
6. Display Options
7. Manipulation Tools
8. Parameters Area
9. Status Bar
10. Apply / Cancel

File Menu for CAM

The File menus in CAM is similar to the menus in CAD, however CAM may offer some Print options.

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

Printing

Page Setup

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

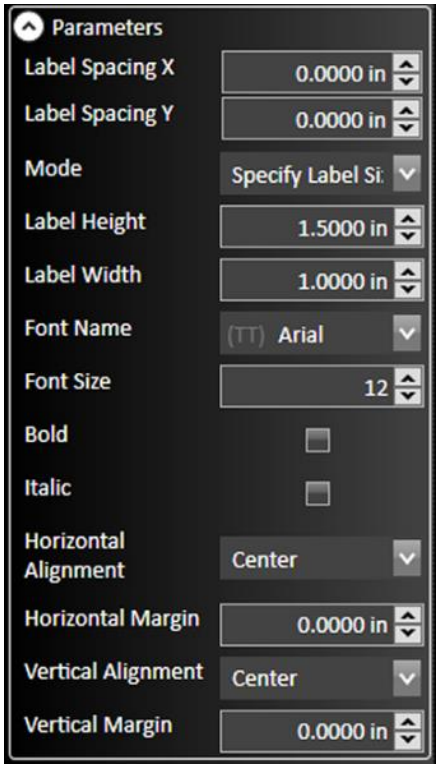
Preview

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

Print

Brings up a Window printer selection window.

Parameters



Label Spacing X/Y

Mode

Select either **Specify Label Size** or **Specify Grid Dimensions**.

Label Height/Width

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

Columns/Rows Number

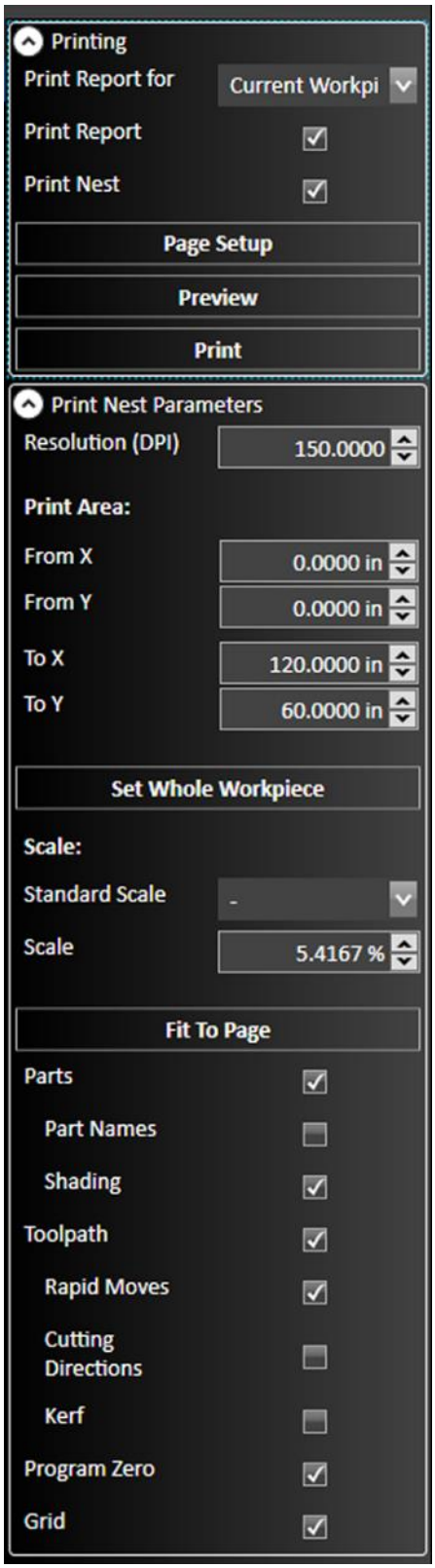
Grid dimensions can be set by assigning the number of columns and rows per sheet of labels.

Font Name/Size/Bold/Italic

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

Horizontal/Vertical Alignment and Margin

Print Nest Report



Print Area

Area to be printed can be set by defining the four corners of a rectangle: From X, To X, From Y, and to Y.

Set Whole Workpiece

Select button to include the whole workpiece in the drawing.

Standard Scale

Select a scale from the drop-down menu: 1:50 to 10:1

Scale

Set scale of drawing as a percentage of actual size.

Fit To Page

Fit drawing to a single page. Automatically calculates scale.

Display Settings

Select which features of the drawing workspace are visible in the print. See Display Options for definitions.

Parts

Part Names

Shading

Toolpath

Rapid Moves

Cutting Directions

Kerf

Program Zero

Grid

CAM Ribbon

The ribbon features an assortment of command icons to define the toolpaths, nest, and generate a G-Code program. The ribbon will show features based on the current document type (2D Part, 3D Part, Pipe)

File

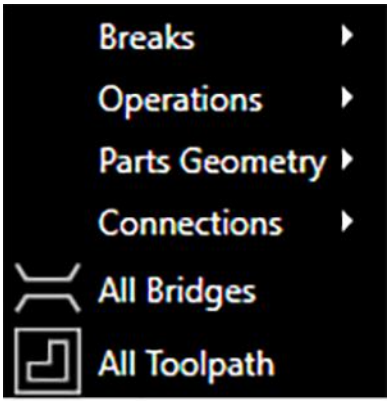
The options are the same as described in the CAD Ribbon.

Feature Type

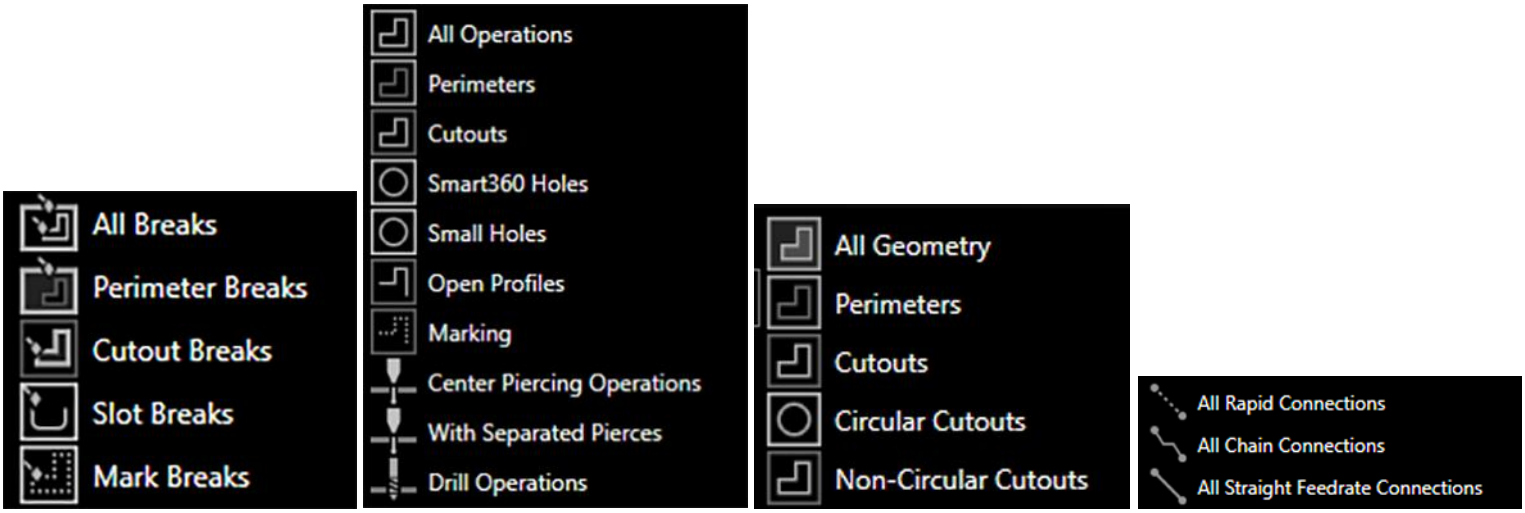
This is for 2D Part Documents. The options are the same as described in the CAD Ribbon.

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** button, a menu of options is displayed: **Breaks**, **Operations**, **Parts Geometry**, **Connections**, **All Bridges**, and **All Toolpath**.



When you select **Breaks** (lead-in/lead-outs), a fly-out menu appears with additional options about specific types of breaks: All Breaks, Perimeter Breaks, Cutout Breaks, Slot Breaks, and Mark Breaks.

When you select **Operations**, a fly-out menu appears with additional options about specific types of operations: All Operations, Perimeters, Cutouts, Smart360 Holes, Small Holes, Open Profiles, Marking, Center Piercing Operations, With Separate Pierces, and Drill Operations.

When you select **Parts Geometry**, a fly-out menu appears with additional options about specific types of geometry: All Geometry, Perimeters, Cutouts, Circular Cutouts, and Non-Circular Cutouts.

When you select **Connections**, a fly-out menu appears with additional options about specific types of connections: All Rapid Connections, All Chain Connections, and All Straight Feedrate Connections.

Settings

The CAM settings ribbon enables you to define the workpiece, define the fabrication head(s) parameters, and other settings that will determine how the toolpath will be generated.

Depending on the features configured on your system and the document type, different settings will be available.

Some features are only available for specific types of fabrication heads (plasma, oxyfuel, waterjet, marker, etc…)

Project Settings

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

General

Default Fabhead

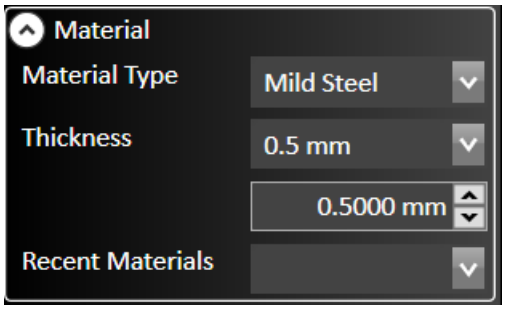
Select the default fabrication head used for each type of toolpath: cutting, marking.

Material Settings

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

Note: The options described here are for flat 2D parts. When working with 3D block parts or pipe/tube parts, a different set of options will be available.

Material



Material Type

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

Thickness

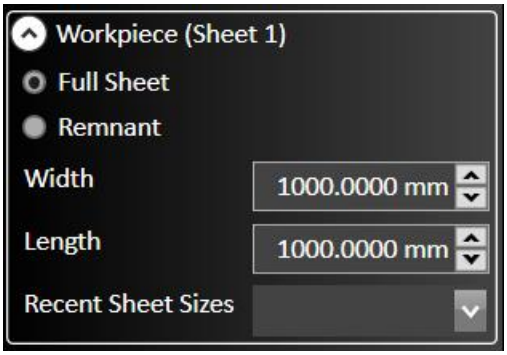
Specify the thickness of your material. The parameters will automatically adjust to match your selection.

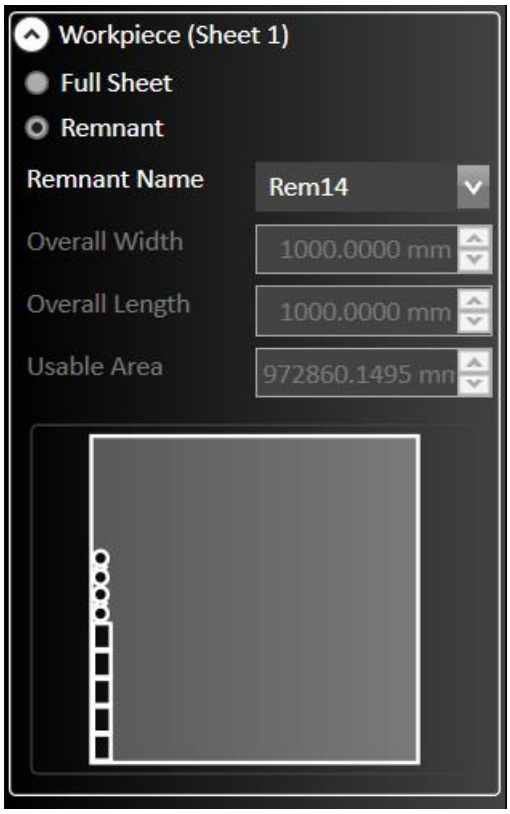
Recent Materials

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

Workpiece

Options vary depending on document type. If your system supports the use of remnants, there will be an option to select either the Full Sheet or Remnant.





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.

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.

Recent Sheet Sizes

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

Remnant Name

Select from a list of existing remnants.

Plasma Settings

If a plasma fabrication head has been configured, its process settings can be adjusted here.

FlashCut offers several parameter settings for various types of plasma torches. Click the icon to bring up options in the parameters window for the plasma model you're configured for.

Process

Here you'll be able to select the **Process** to use. For example: Cut, Mark, Small Hole, Smart360, or a custom Process that you created.

Then you can select parameters that act as inputs for a cut chart lookup. The available parameters depend on the type of plasma torch that is configured.

In general, parameters located above this **Lookup/Reset to Defaul**t button are inputs for the cut chart lookup and settings located below this button are outputs from the cut chart lookup.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.

Depending on the type and model of plasma system that is configured, different plasma settings will be available for that specific system.

AutoCut / UltraCut

Tip

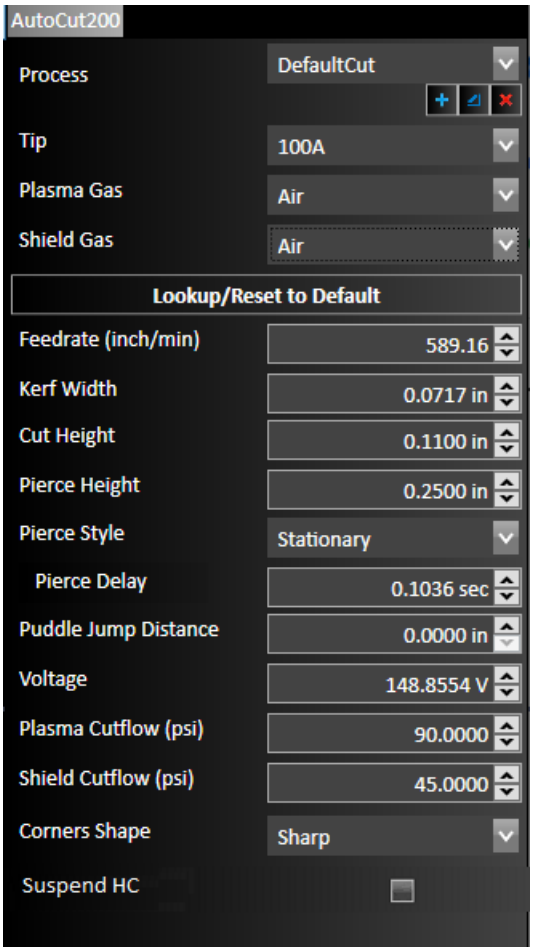
Select your plasma tip model. The parameters will automatically adjust to match your selection.

Plasma Gas

Select the plasma gas.

Shield Gas

Select the shield gas.



Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Plasma Cutflow

Specifies the flow rate of the plasma gas.

Shield Cutflow

Specifies the flow rate of the shield gas.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Cutmaster

Tip

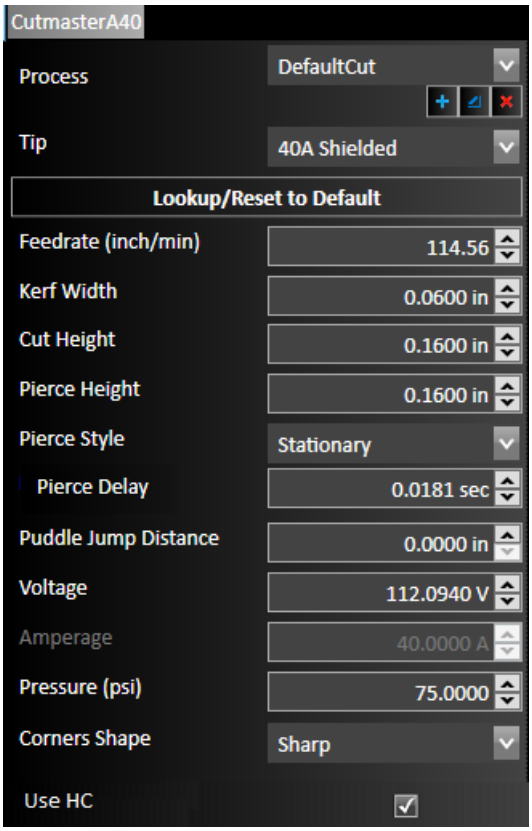
Select your plasma tip model. The parameters will automatically adjust to match your selection.

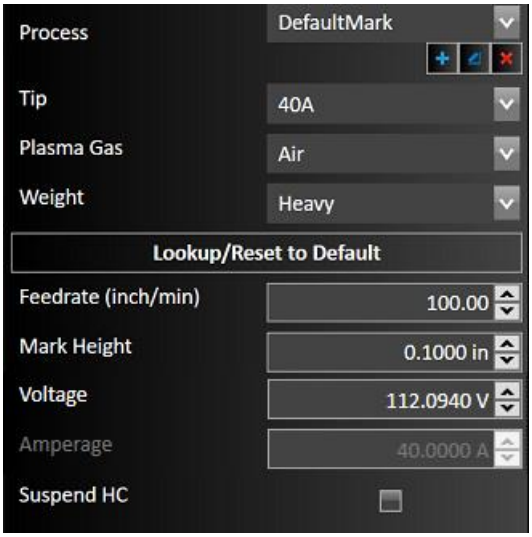
Plasma Gas

Select the plasma gas.

Weight

Select the mark weight.





Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Pressure

Specifies the air pressure of the torch.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Lincoln FlexCut

Nozzle

Select your plasma nozzle model. The parameters will automatically adjust to match your selection.

Plasma Gas

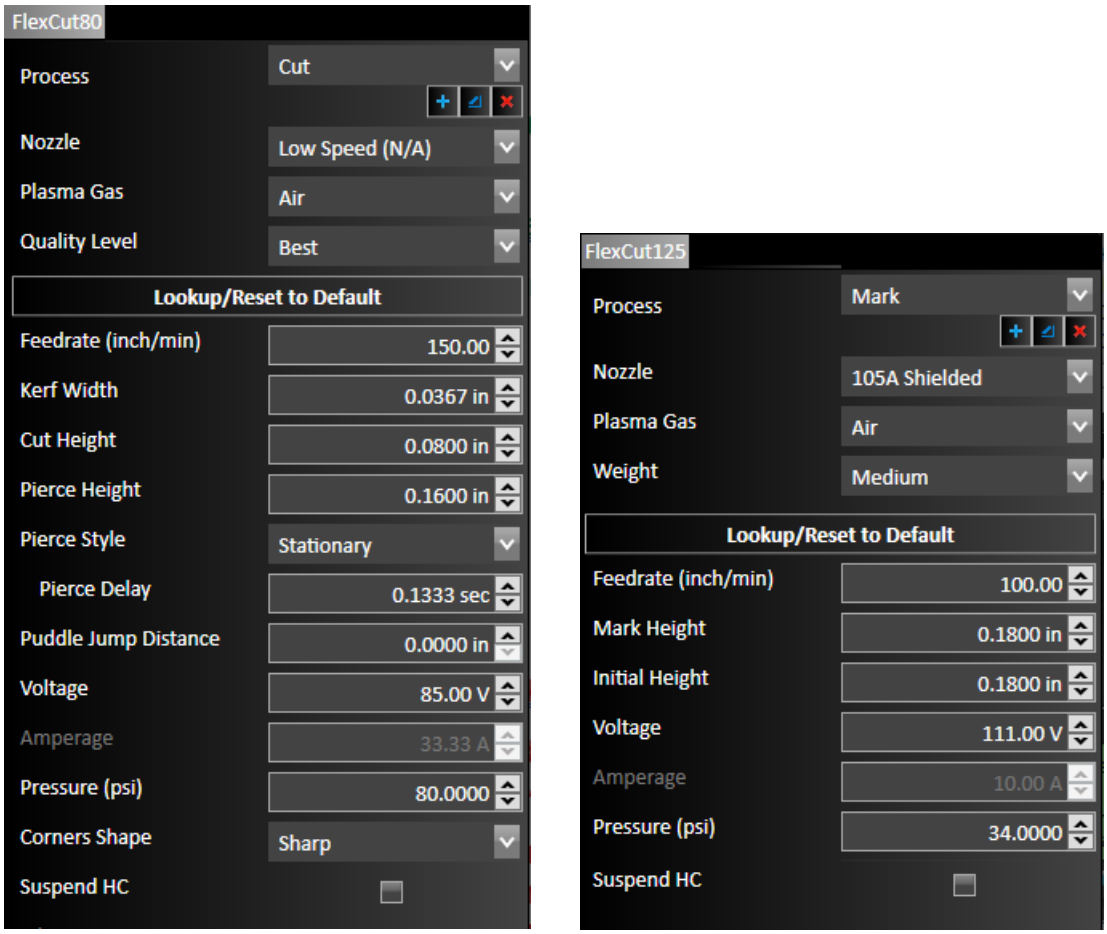
Select the plasma gas.

Quality Level

Select whether the toolpath will be optimized for best quality or fast production.

Weight

Select the mark weight.



Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Initial Height

The distance the plasma torch is above the material to initiate marking.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Pressure

Specifies the air pressure of the torch.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Hypertherm HyPerformance HPR

Nozzle

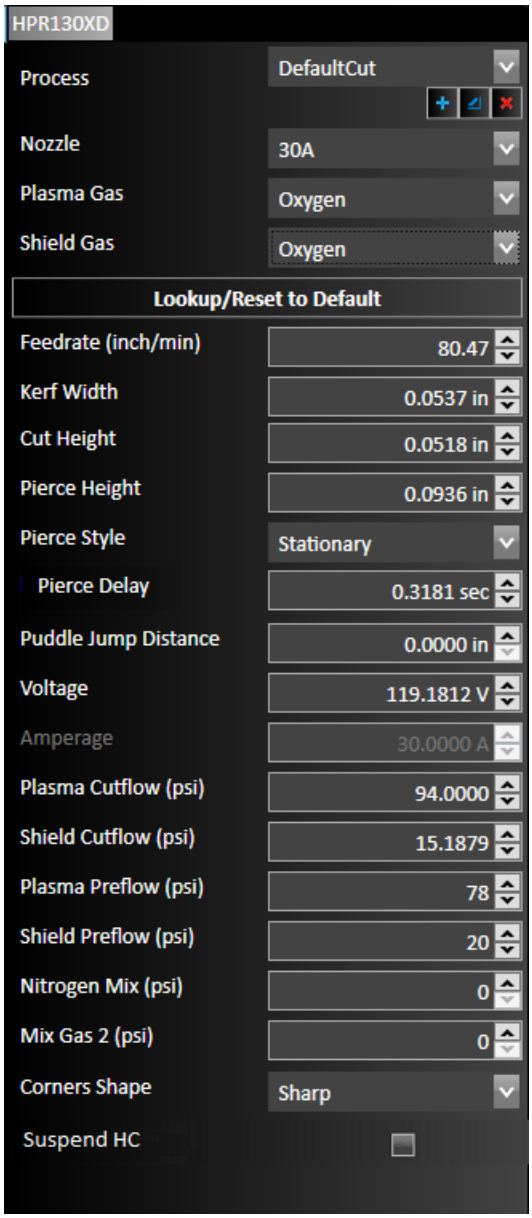
Select your plasma nozzle model. The parameters will automatically adjust to match your selection.

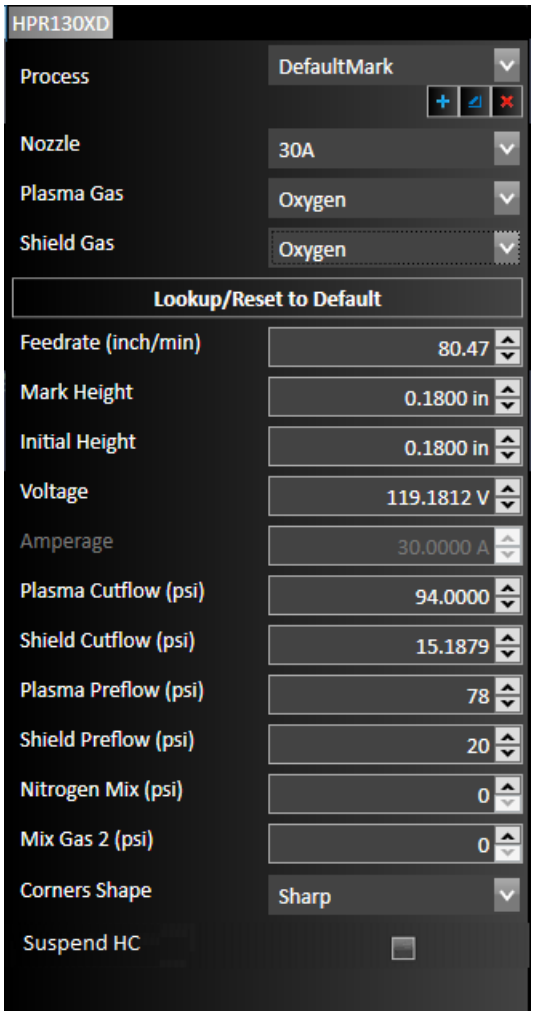
Plasma Gas

Select the plasma gas.

Shield Gas

Select the shield gas.





Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Initial Height

The distance the plasma torch is above the material to initiate marking.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Plasma Cutflow

Specifies the pressure of the plasma gas.

Shield Cutflow

Specifies the pressure of the shield gas.

Plasma Preflow

Specifies the pressure of the plasma gas.

Shield Preflow

Specifies the pressure of the shield gas.

Nitrogen Mix

Specifies the pressure of the nitrogen mix gas.

Gas 2 Mix

Specifies the pressure of the gas 2 mix gas.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Hypertherm HySpeed HSD

Nozzle

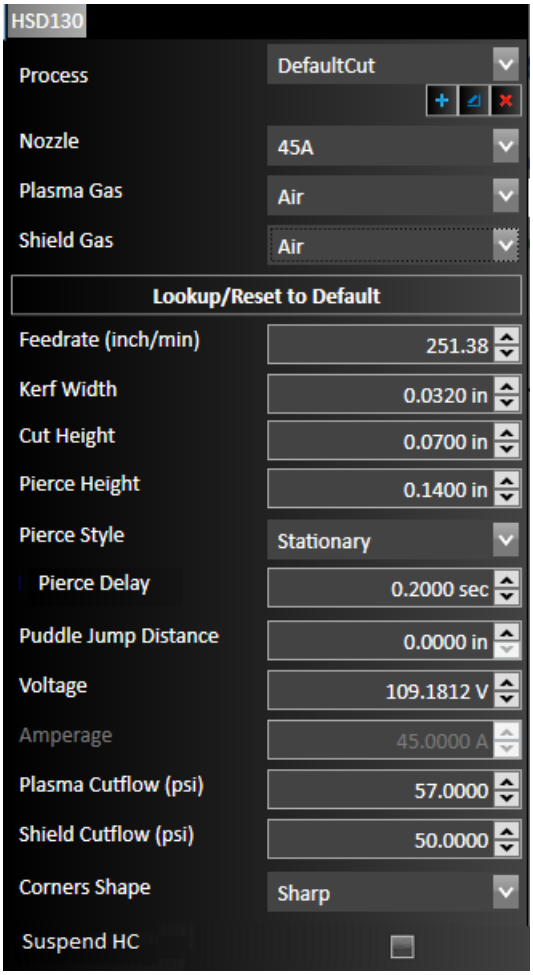
Select your plasma nozzle model. The parameters will automatically adjust to match your selection.

Plasma Gas

Select the plasma gas.

Shield Gas

Select the shield gas.



Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Plasma Cutflow

Specifies the pressure of the plasma gas.

Shield Cutflow

Specifies the pressure of the shield gas.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Kjellberg Cutfire / HiFocus / SmartFocus

Technology

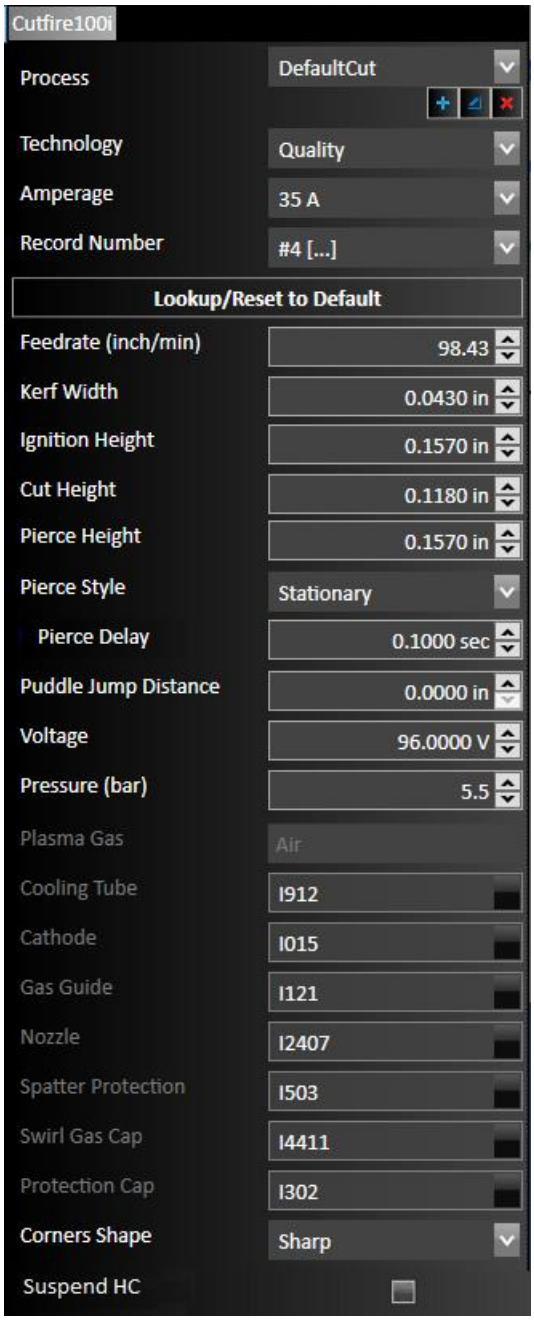
Select your technology, such as Contour Cut, Contour Cut Speed, Fine Cut. The parameters will automatically adjust to match your selection.

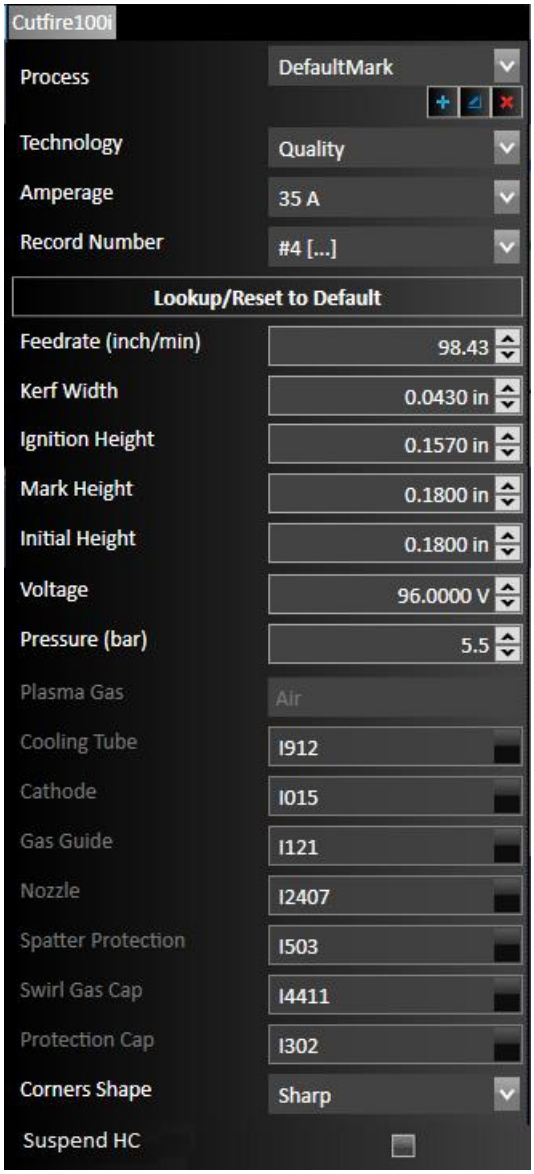
Amperage

Select from the list of available based on your Technology.

Record Number

Select from a list of the pertinent record numbers.





Feedrate

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

Kerf Width

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

Ignition Height

The distance the plasma torch is above the material to ignite the torch.

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Initial Height

The distance the plasma torch is above the material to initiate marking.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Plasma Gas 1

This is a non-editable field.

Plasma Gas 2 & Pressure

The gas type is non-editable.

Swirl Gas 1 & Pressure

The gas type is non-editable.

Swirl Gas 2 & Pressure

The gas type is non-editable.

Shield Preflow

Specifies the pressure of the shield gas.

Time Up/Down Slope

Specifies the up/down slope time in milliseconds.

Gas / Special Code

This is non-editable values of the gas code and special code.

Part Numbers

Listed will be non-editable fields of the part numbers of the Cooling Tube, Cathode, Gas Guide, Nozzle, Nozzle Cap, Swirl Gas Cap, Protection Cap.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

MaxPro

Nozzle

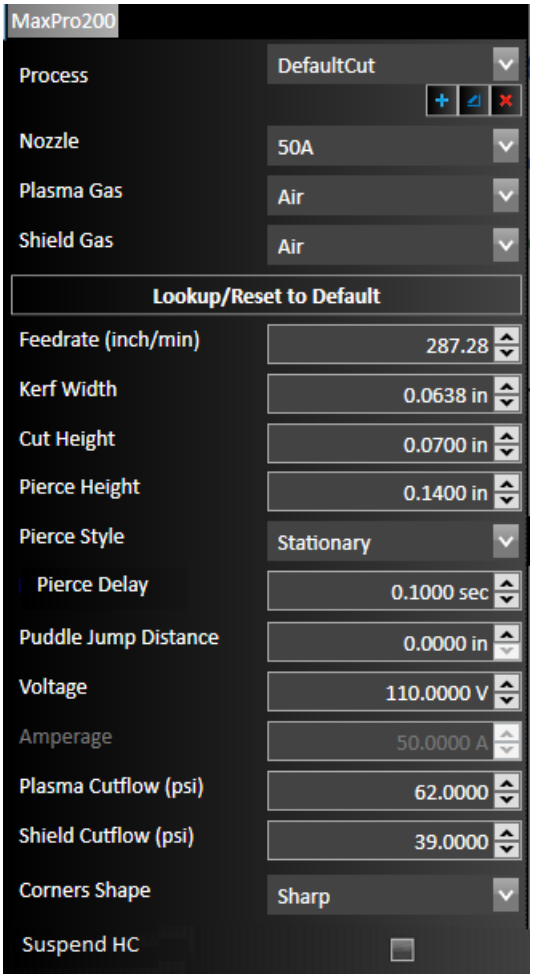
Select your plasma nozzle model. The parameters will automatically adjust to match your selection.

Plasma Gas

Select the plasma gas.

Shield Gas

Select the shield gas.



Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Plasma Cutflow

Specifies the pressure of the plasma gas.

Shield Cutflow

Specifies the pressure of the shield gas.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Hypertherm Powermax

Nozzle

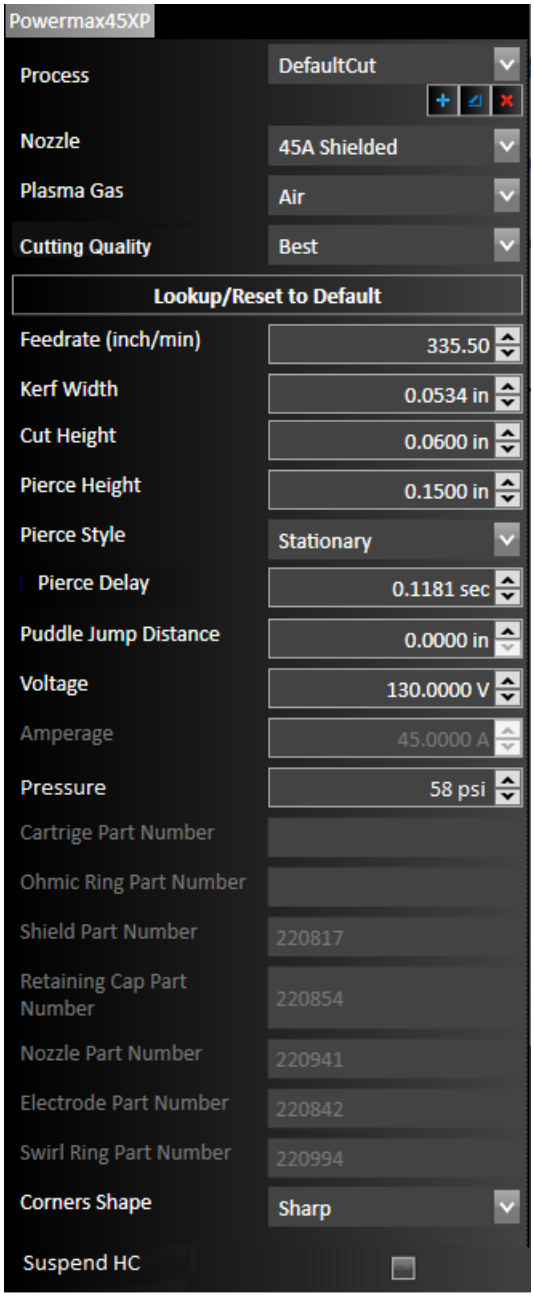
Select your plasma nozzle model. The parameters will automatically adjust to match your selection.

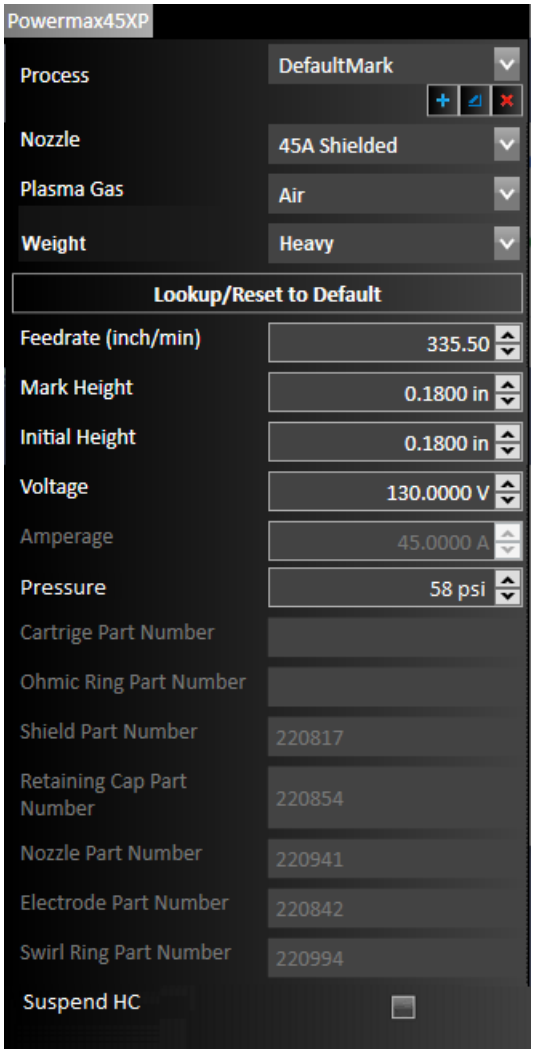
Plasma Gas

Select the plasma gas.

Cutting Quality

Select whether the toolpath will be optimized for best quality or fast production.





Feedrate

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

Kerf Width

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

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Initial Height

The distance the plasma torch is above the material to initiate marking.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

This is a non-editable field that displays the amperage of the torch.

Pressure

Specifies the air pressure of the torch.

Part Numbers

Listed will be non-editable fields of the part numbers, depending on plasma torch model and process, of the Cartridge, Ohmic Ring, Shield, Retraining Cap, Nozzle, electrode, Swirl Ring.

Suspend HC

When checked, will suspend automatic height control.

Edge Start

This is a non-editable indicator based on the cut chart. When checked, an Edge Start is recommended.

Hypertherm XPR

Amperage

Select from the list of available based on your model.

Cutting Gases

Select the plasma cutting gases.

Cutting Type

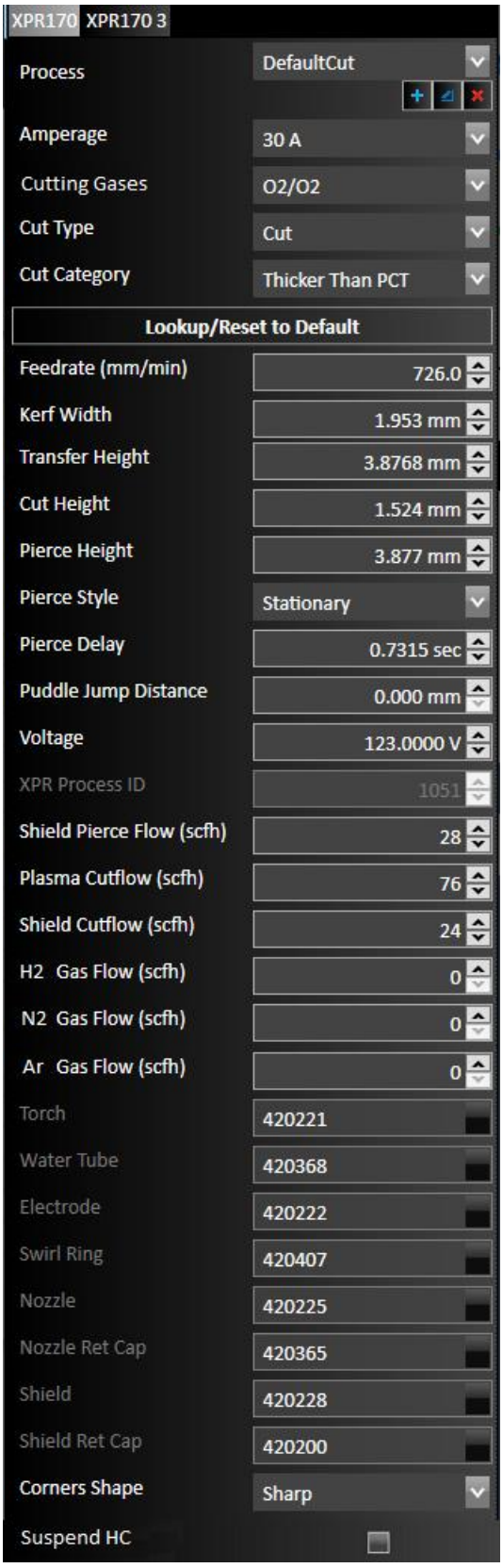
Select the cut type. Cut, Underwater.

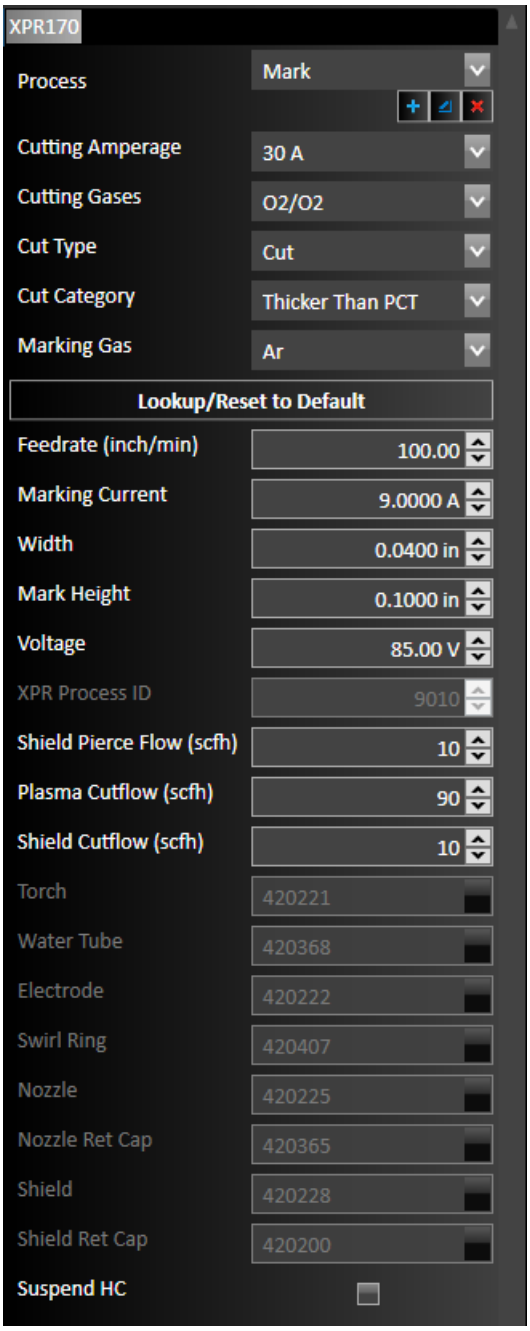
Cut Category

Select the cut category. Process Core Thickness, Thicker Than Percent, Thinner Than Percent, Edge Start Only, Severance.

Marking Gas

Select the plasma marking gas.





Feedrate

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

Kerf Width

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

Transfer Height

The distance the plasma torch is above the material to start arc transfer for cutting.

Cut Height

The distance the plasma torch is above the material while cutting.

Mark Height

The distance the plasma torch is above the material while marking.

Marking Current

Specify the mark current.

Pierce Height

The distance the plasma torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For plasma, this can be a regular stationary pierce or an edge start.

Pierce Delay

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

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

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 will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

XPR Process ID

This is a non-editable field that displays the XPR process ID.

Shield Pierce Flow

Specifies the flow rate of the shield pierce gas in standard cubic feet per hour.

Plasma Cutflow

Specifies the flow rate of the plasma cut gas.

Shield Cutflow

Specifies the flow rate of the shield cut gas.

H2 Gas Flow

Specifies the flow rate of the hydrogen gas.

N2 Gas Flow

Specifies the flow rate of the nitrogen gas.

Ar Gas Flow

Specifies the flow rate of the argon gas.

Part Numbers

Listed will be non-editable fields of the part numbers of the Torch, Water Tube, Electrode, Swirl Ring, Nozzle, Nozzle Retaining Cap, Shield, Shield Retaining Cap.

Suspend HC

When checked, will suspend automatic height control.

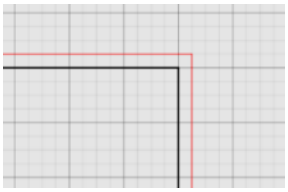
Corners Shape

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.

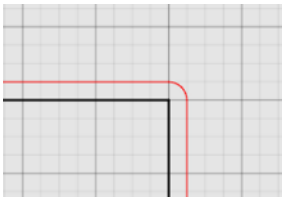
Sharp corners

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



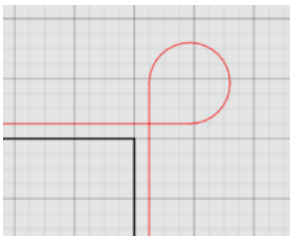
Rounded corners

The toolpath describes an arc at the corner of the part, maintaining a consistent distance from the edge.



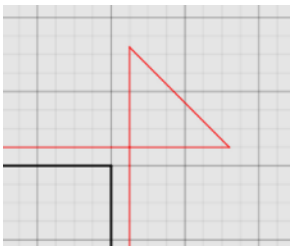
Rounded loops

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.



Small Hole Process

If your system does not include Smart360, you may want to use the Small Hole process for cutting holes.

Diameter to Thickness Radio <=

This defines the size of holes that will be cut with this process. Holes equal to or smaller than this diameter will be included.

Reduce Feedrate to

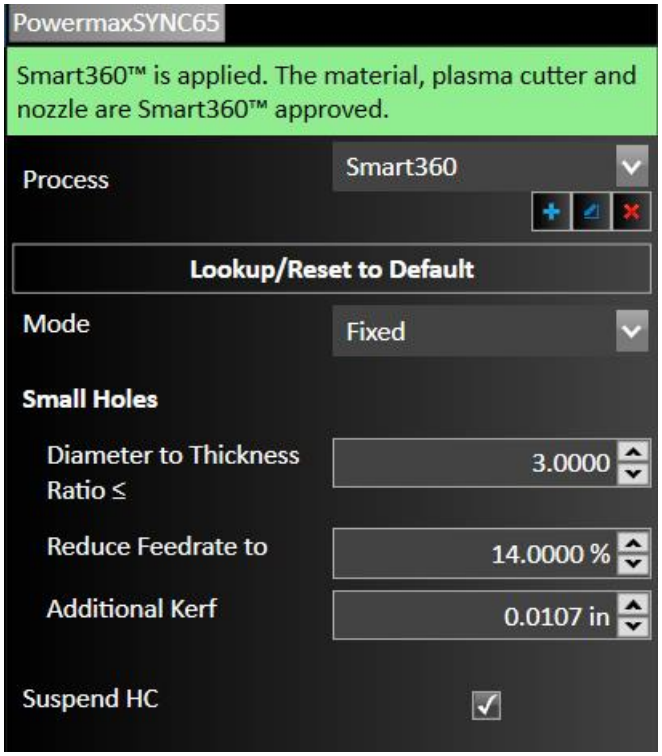
This is a percentage of the feedrate used for cutting the hole, based on the regular cutting process. For example, if the regular cutting process has a feedrate of 100 in/min and this value is 14%, the hole cutting process feedrate will be 14 in/min.

Suspend HC

This enables or disables automatic height control. In most cases, it is recommended to suspend automatic height control because it may not behave in a desired way while cutting a hole at a reduced feedrate.

Smart360™ Process

This Process creates high accuracy holes and small cutouts by manipulating torch, kerf, and feedrate settings. The options available will depend on the plasma model you're using.



Mode

Fixed mode applies the same reduced feedrate and additional kerf for all holes that use the Smart360 process. Adaptive mode will vary the reduced feedrate and additional kerf for holes that use the Smart360 process based on the hole diameter to material thickness ratio.

Diameter to Thickness Radio <=

This defines the size of holes that will be cut with this process. Holes equal to or smaller than this will apply.

Reduce Feedrate to (Fixed Mode)

1:1 Ratio Feedrate (Adaptive Mode)

This is a percentage of the feedrate used for cutting the hole, based on the regular cutting process. For example, if the regular cutting process has a feedrate of 100 in/min and this value is 14%, the hole cutting process feedrate will be 14 in/min.

Additional Kerf (Fixed Mode)

1:1 Ratio Additional Kerf (Adaptive Mode)

When cutting at a reduced feedrate the kerf can be wider than at a full feedrate. This value will be added to the regular cutting process kerf and the toolpath will be offset accordingly for a more accurate hole size.

Suspend HC

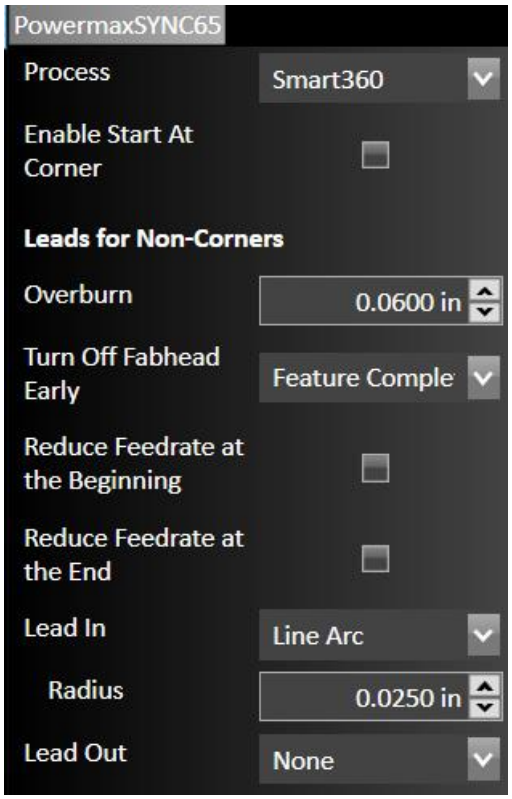
This enables or disables automatic height control. In most cases, it is recommended to suspend automatic height control because it may not behave in a desired way while cutting a hole at a reduced feedrate.

Smart360 Kjellberg

When using some Kjellberg models of plasma systems, such as HiFocus or SmartFocus, you will have a unique Smart360 process available. This process will automatically calculate the feedrate, kerf, and cut height based on Kjellberg cut chart data and the ratio of the hole diameter to material thickness.

Break Settings for Smart360

Unique break settings can be configured for each Process. Smart360 typically uses a slight Overburn and a Line-Arc Lead-In.

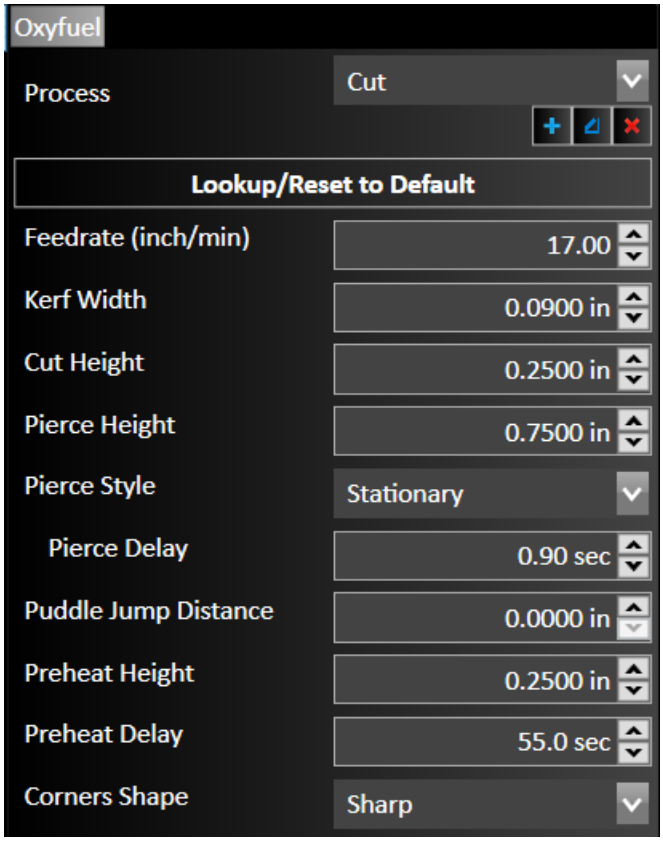

PowermaxSYNC65
Process
Smart360
Enable Start At Corner
Leads for Non-Corners
Overburn
0.0600 in
Turn Off Fabhead Early
Feature Comple
Reduce Feedrate at the Beginning
Reduce Feedrate at the End
Lead In
Line Arc
Radius
0.0250 in
Lead Out
None

Oxyfuel Settings

If an oxyfuel fabrication head has been configured, its settings can be adjusted here.

FlashCut offers several parameter settings for various configurations of oxyfuel cutting systems.

You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes.



Feedrate

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

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.

Cut Height

The distance the waterjet is above the material while cutting.

Pierce Height

The distance the oxyfuel torch is above the material for piercing.

Pierce Style

Specify how to pierce the material. For oxyfuel, this can be a regular stationary pierce, an edge start, or creep where you also specify the creep rate and time.

Puddle Jump Distance

This is the horizontal distance to move after the pierce is complete. The lift axis will hold at the pierce height then go down to the cut height after the puddle jump distance.

Preheat Height

This is height the oxyfuel torch will be over the material while preheating.

Preheat Delay

This is the time to preheat the material before cutting.

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.

Waterjet Settings

If a waterjet fabrication head has been configured, its settings can be adjusted here.

FlashCut offers several parameter settings for various configurations of waterjet cutting systems.

Process

Here you'll be able to select the **Process** to use. For example: Cut, Small Hole, or a custom Process that you created.

Then you can select parameters that act as inputs for lookup. The available parameters depend on what features are configured for your waterjet cutting system.

In general, parameters located above this **Lookup/Reset to Defaul**t button are inputs for lookup and settings located below this button are outputs from the lookup.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.

Nozzle / Orifice Diameter

Nozzle / Orifice diameter for the cutting head.

Machinability

This is a non-editable value that is automatically calculated based on the material.

Use Abrasive

Abrasive Flow Rate and Factor

Flow rate and abrasiveness of the abrasive compound.

Cutting Quality

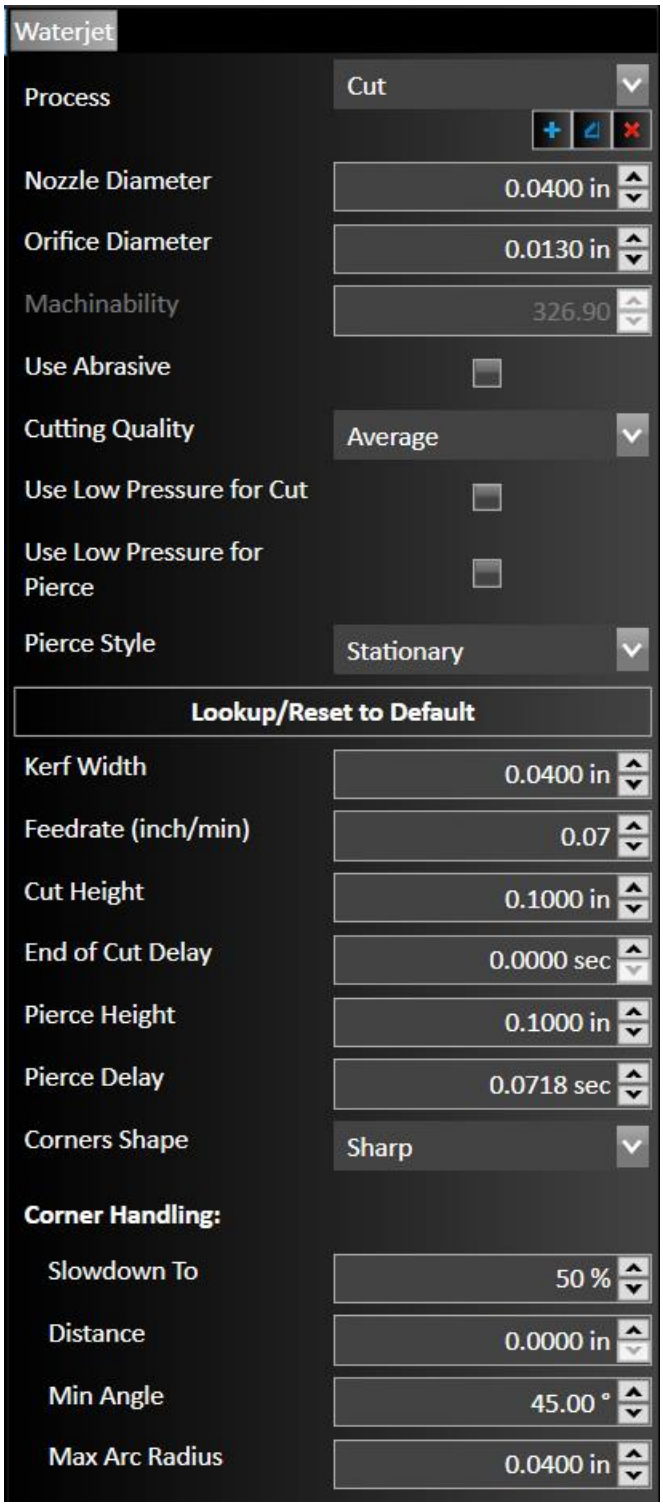
Low, fair, average, good, or excellent

Use Low Pressure for Cut / Pierce

Support for Low Pressure Mode must be enabled in config file.

Pierce Style

Specify how to pierce the material. For waterjet, this can be a regular stationary pierce, an edge start, creep, wiggle, or circular.



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.

Feedrate

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

Cut Height

The distance the waterjet is above the material while cutting.

End of Cut Delay

The time to leave the waterjet on after getting to the end of the toolpath.

Pierce Height

The distance the waterjet is above the material for piercing.

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.

Corner Handling

The feedrate the waterjet cuts can be slowed down at corners or small arcs of the toolpath.

Slowdown To - Specify how much to slowdown to by a percentage of the feedrate.

Distance - Specify the distance before the corner or arc to begin slowing down.

Min Angle - Specify the minimum angle of corner, relative to the direction of travel along the toolpath, for which to slowdown. For slight corners, less than this angle, there will be no slowdown. For sharp corners, greater than this angle, there will be a slowdown.

Max Arc Radius - Specify the maximum radius of the arc for which to slowdown. For large arcs, greater than this radius, there will be no slowdown. For small arcs, less than this radius, there will be a slowdown.

Abrasive On Delay

The time after turning on the water before turning on the abrasive. Set this to a negative value if you want the abrasive to turn on before the water is turned on.

Abrasive Off Delay

The time after turning off the abrasive before turning off the water. Set this to a negative value if you want the water to turn off before the abrasive is turned off.

Laser Settings

If a laser cutter fabrication head has been configured, its settings can be adjusted here.

FlashCut offers several parameter settings for various configurations of laser cutting systems.

Process

Here you'll be able to select the **Process** to use. For example: Cut, Small Hole, or a custom Process that you created.

The available parameters depend on what features are configured for your waterjet cutting system.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.

Marker Settings

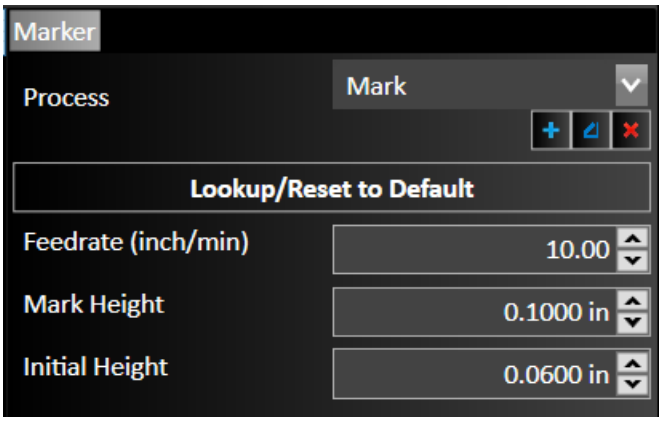
If a marker / scribe fabrication head has been configured, its settings can be adjusted here.

Process

Here you'll be able to select the **Process** to use. For example: Mark or a custom Process that you created.

The available parameters depend on what features are configured for your marking system.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.



Feedrate

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

Mark Height

The distance the marker is above the material while marking.

Initial Height

The distance the marker is above the material to initiate marking.

Spindle Settings

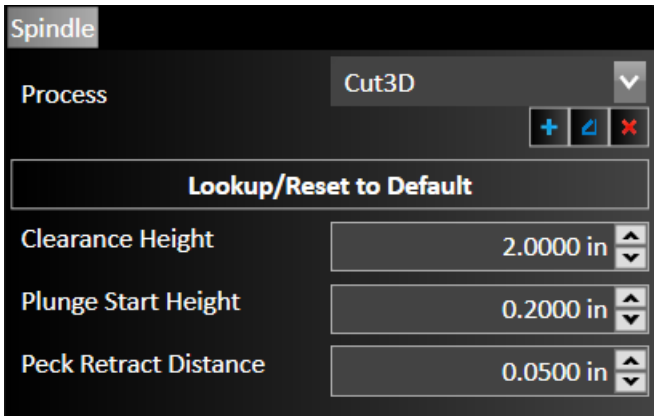
If a spindle / mill fabrication head has been configured, its settings can be adjusted here.

Process

Here you'll be able to select the **Process** to use. For example: Cut3D or a custom Process that you created.

The available parameters depend on what features are configured for your system.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.



Clearance Height

Height of the spindle during rapid moves.

Plunge Start Height

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

Peck Retract Distance

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

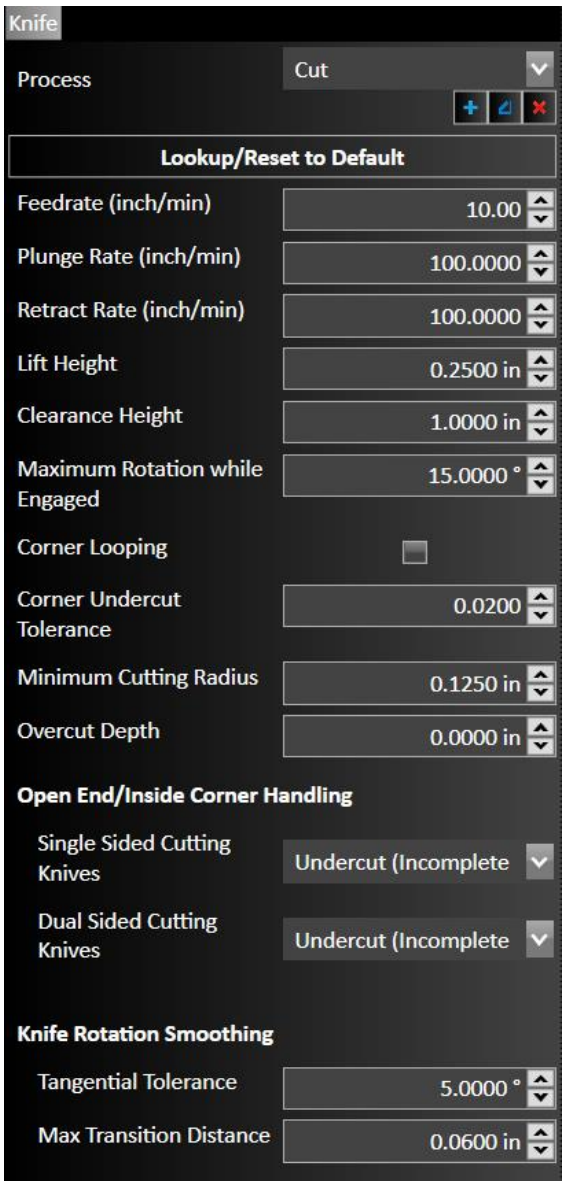
Knife Settings

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

Process

Here you'll be able to select the **Process** to use. For example: Cut3D or a custom Process that you created. The available parameters depend on what features are configured for your system.

You can override several of the recommended process settings by simply typing in the respective field. After specifications are complete, you may either **Apply** or **Cancel** the changes for this document.



Feedrate

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

Plunge Rate/Retract Rate

Rate during negative lift axis movements.

Lift Height

Height of cutting head above workpiece during cuts.

Maximum Rotation while Engaged

Degrees of rotation permitted.

Corner Looping

When selected a toolpath will be generated in a way that moves the knife in a loop around corners. This can result in more accurate corners of the part. Options are Rounded Loops or Triangular Loops.

Extension Distance – how far beyond corner the corner to extend the loop.

Maximum Angle to Add Loop – largest angle for which a loop toolpath will be generated.

Corner Undercut Tolerance

Minimum Cutting Radius

Overcut Depth

This is the distance to cut through the material.

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)

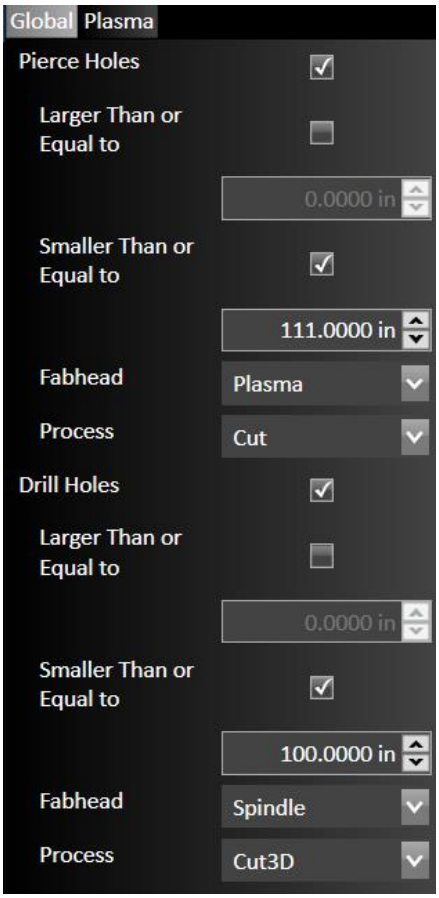
Knife Rotation Smoothing

Tangential Tolerance

Max Transition Distance

Hole Treatment Settings

This tool is available when the installed fabrication head defines hole radius ranges for automatic Drill or Pierce operations. An example from a Plasma fabrication head is shown below.



Pierce Holes

Turns the feature on or off, defines the upper and lower bounds of the hole radius. Available for fabrication heads that are configured for standalone piercing, such as plasma.

Drill Holes

Turns the feature on or off, defines the upper and lower bounds of the hole radius. Available for mill/spindle fabrication heads.

Change Project Break Settings

Use the Project Break 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,

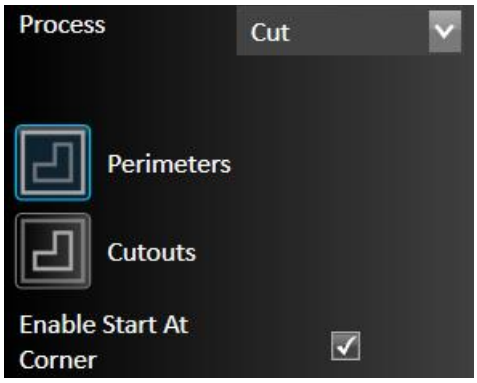
Perimeters

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

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.



Smart 360™

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

Enable Start At Corner

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

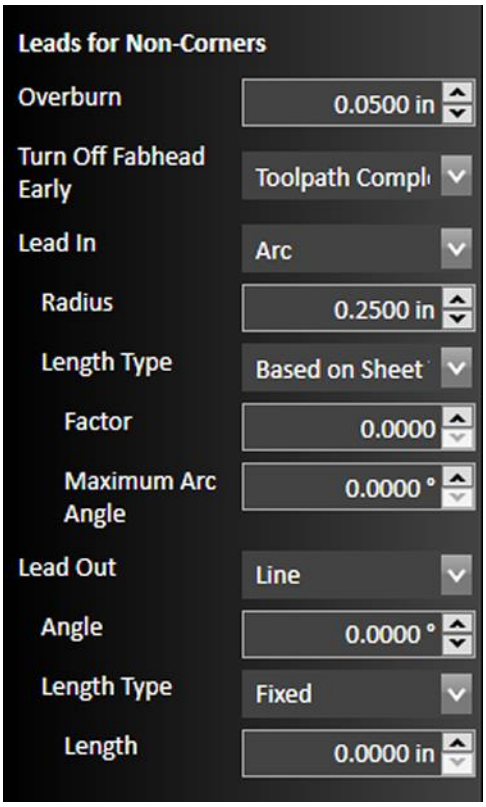
- **Min/Max Corner Angle**

  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 within the 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.

Leads for Non-Corners

Sets the default pattern for leads.



Overburn – Set distance. To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. Lead out lines appear after the overburn.

Turn Off Fabhead Early

If desired, the Fab Head can be turned off at different points 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

Fabhead Off Distance field displayed.

After Feature Completion

Fabhead 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.

Length Type options – **Length Fixed** or **Based on Sheet Thickness**

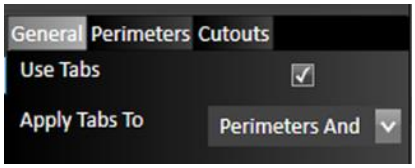
None – No lead will be used.

Change Project Tab Settings

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

Use Tabs

When selected, tabs will be included in the toolpath for the drawing.

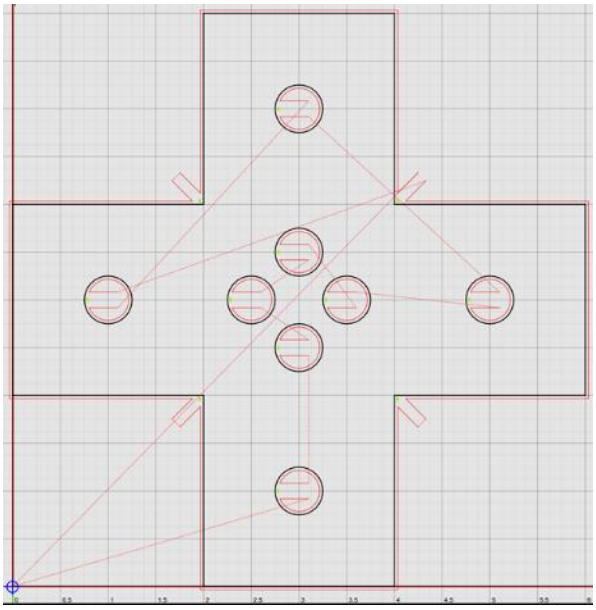


Apply Tabs

Tab settings can be configured for perimeters and cutouts, either separately or together. Three labeled tabs enable detailed configuration.

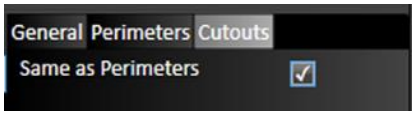
In this drawing, four perimeter tabs have been used to keep the part connected to the workpiece. The tabs appear at the right-angle corners formed by the arms of the cross.

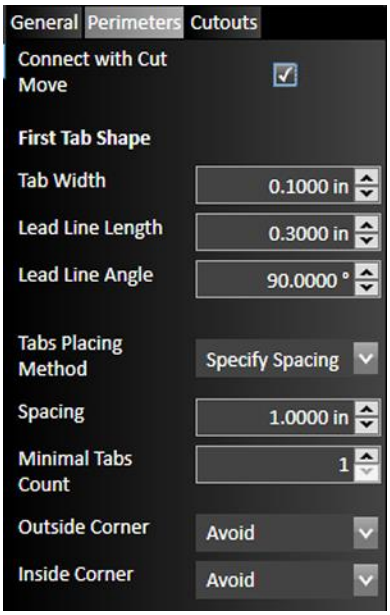
In addition, one tab has been defined for each cutout, keeping the disks inside each hole attached to the part.



Same as Perimeters

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





Connect with Cut Move

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

First Tab Shape Tab Width

Define tab width.

Lead Line Length

Define tab length.

Lead Line Angle

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

Tabs Placing Method

Specify Spacing places tabs by defining the space between them.

Spacing

Minimal Tabs Count

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

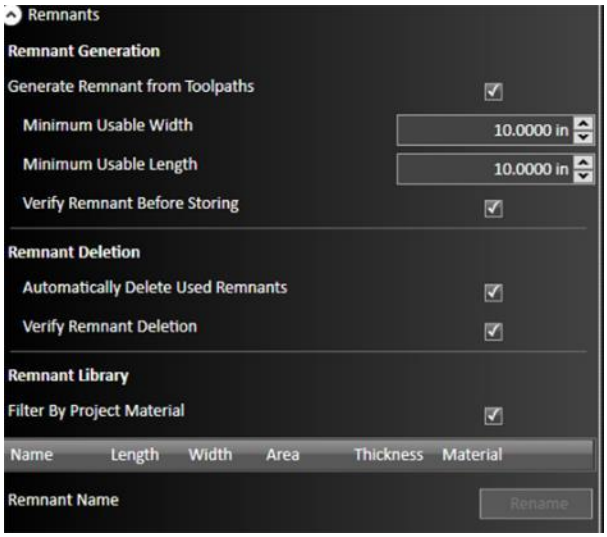
Count of Tabs

Outside/Inside Corner

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

Manage Remnants

Manage remnants can calculate the space remining in a partially used workpiece, and add that workpiece to a library of remnants so that they can be used again.When active, the user is prompted for information about the remnant(s) before leaving the CAM window.



Generate Remnant from Toolpaths

Check to track remnants.

Minimum usable width/length

Sets size limits for tracking remnants of workpieces.

Verify Before Storing

Check to be prompted for information when finished.

Automatically Delete Used Remnants

Check to remove used remnants from the library.

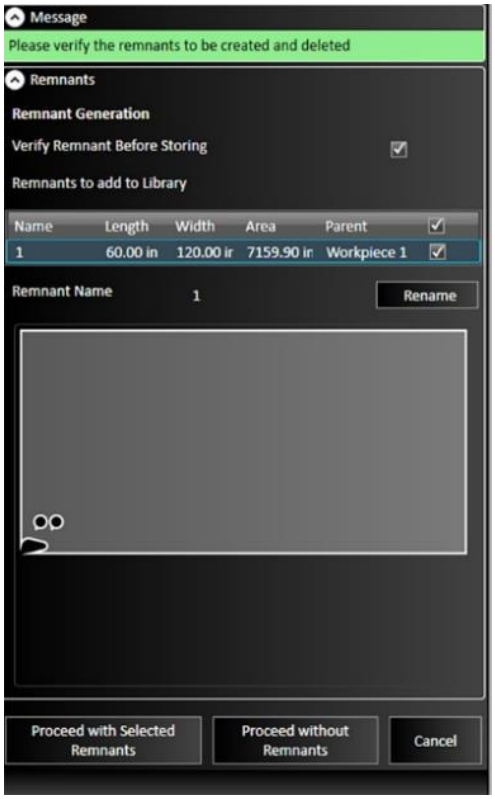
Verify Remnant Deletion

Check to be prompted before remnants are deleted.

Remnant Library

Each remnant can be assigned a name. Other information is automatically generated from known information about the workpiece.

An example of the verification dialog is shown at right.



CAM Actions

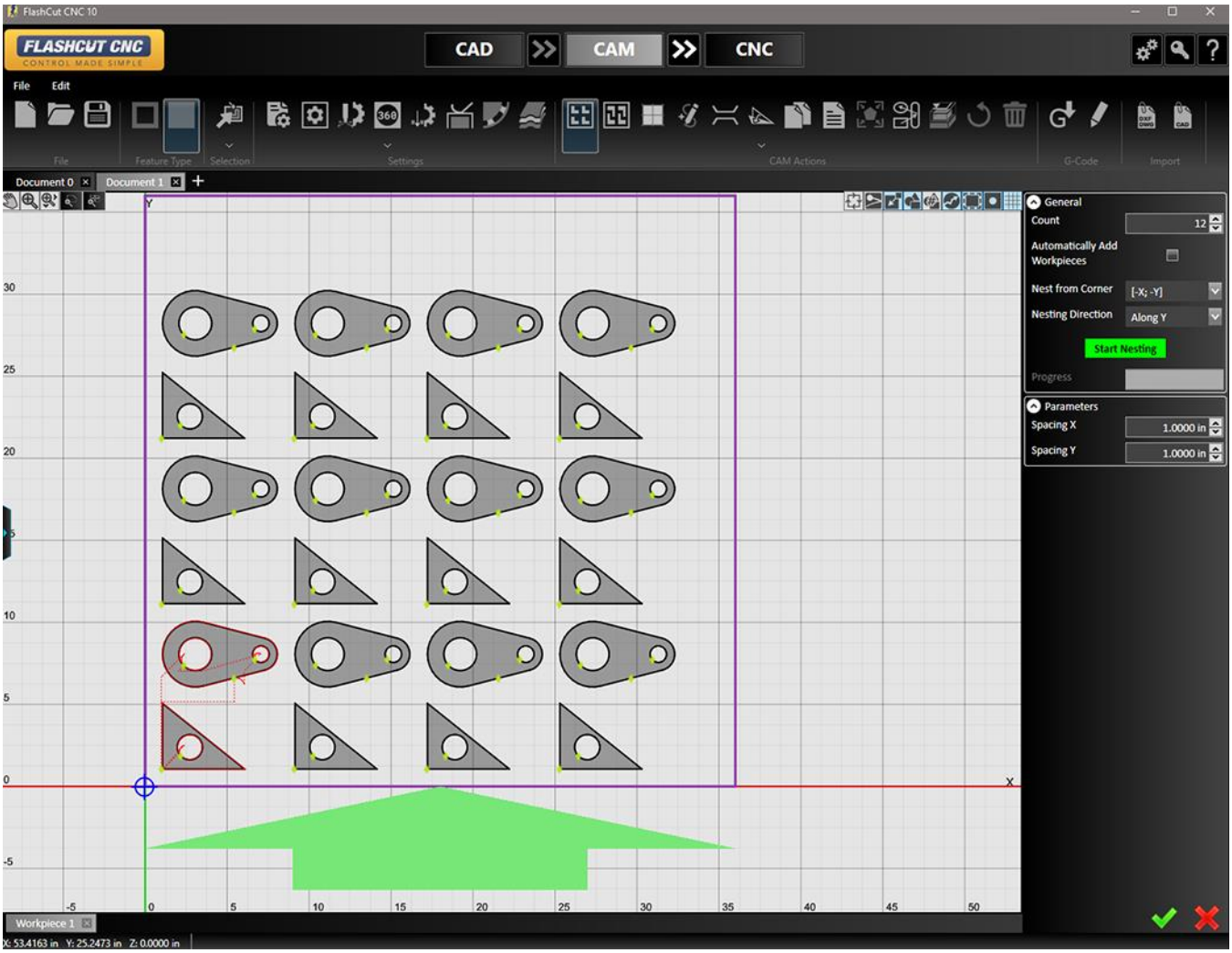
The CAM actions bar enables you to replicate and nest various features in the CAM drawing, as well as select various settings for lead in and lead out lines and the plasma torch. Clicking each option brings up numerous settings in the parameter window.

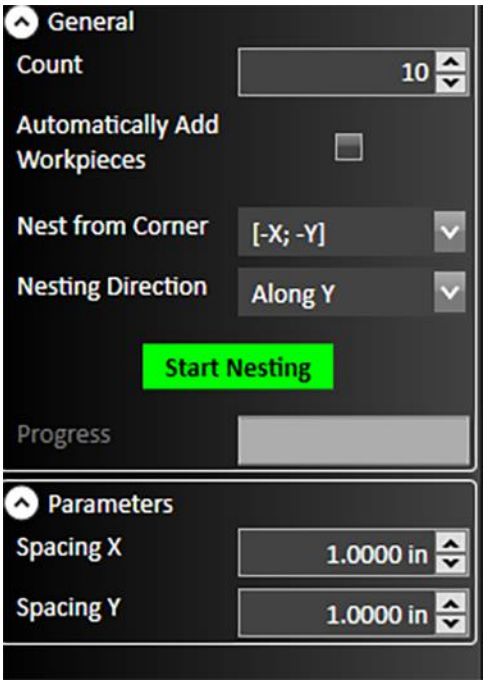
Some icons are only visible for specific types of Fab Heads. For example, Smart360™ settings are only visible if licensed for a plasma Fab Head.

Grid Nesting

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

Copies fill rows left to right, and when the edge of the material is reached, they will advance up a row. Grid nesting does not change the orientation of any of the parts being nested.





Count

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

Automatically Add Workpieces

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

Nest from corner

Selects the starting point of the nesting operation.

Nesting direction

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

Start Nesting

Starts the nesting operation.

Progress

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

Spacing X

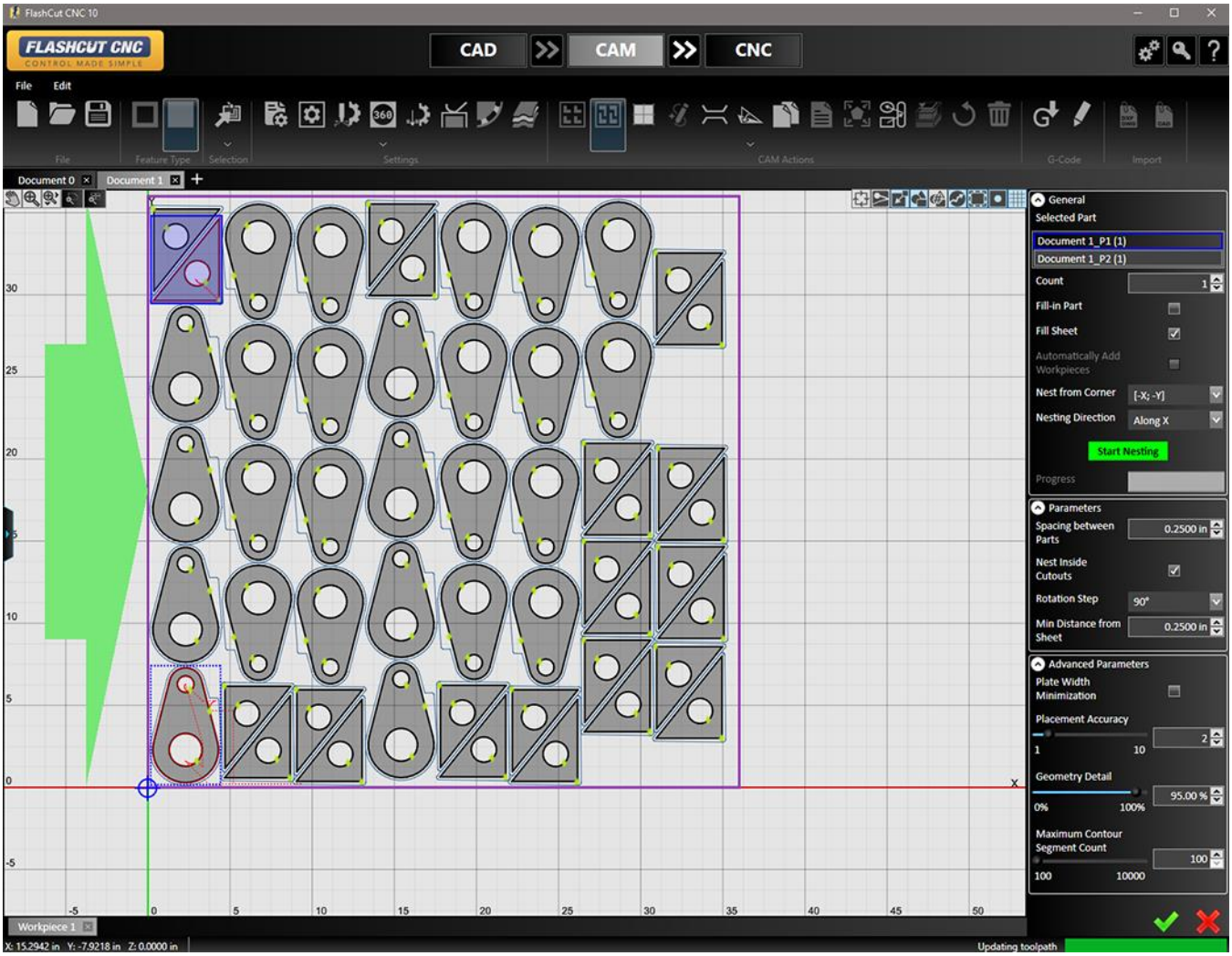
Sets the horizontal spacing between parts.

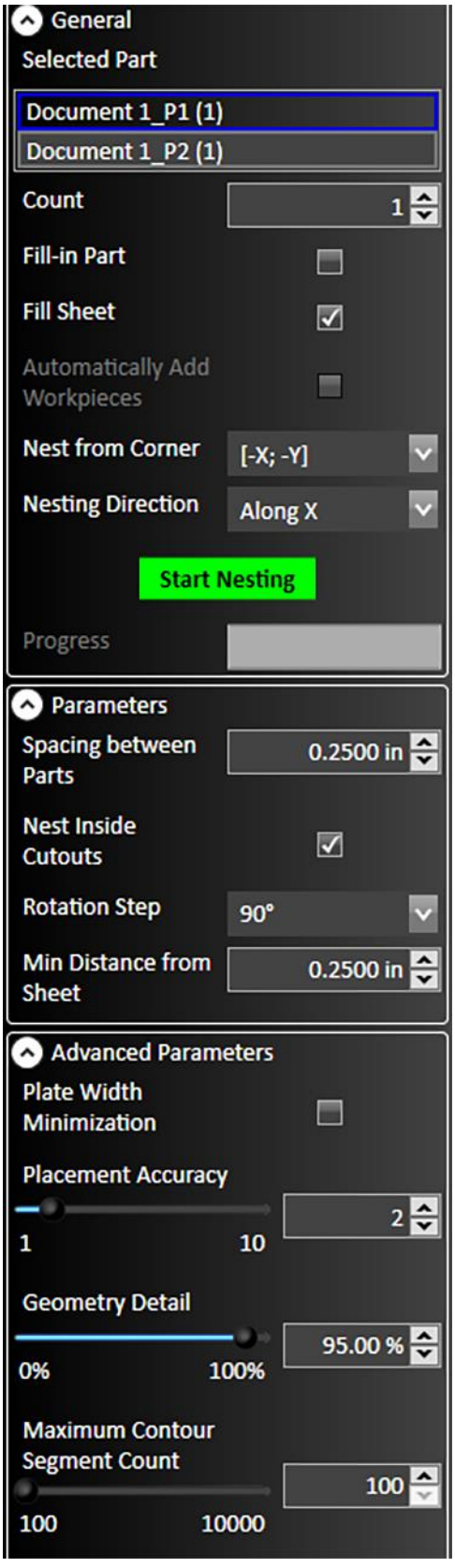
Spacing Y

Sets the vertical spacing between parts.

TrueShape Nesting

The TrueShape nesting function will duplicate, rotate, and translate each part to create a nest that minimizes the amount of material used in a sheet.





Selected Part

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

Count

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

Fill In Part

Visible when multiple parts are available. When selected, the selected part will be added to fill the workpiece. Only one part can be selected.

Fill Sheet

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

Automatically Add Workpieces

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

Nest from Corner

Selects the starting point of the nesting operation.

Nesting Direction

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

Start Nesting

Starts the nesting operation.

Progress

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

Spacing between Parts

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

Nest Inside Cutouts

When selected, parts will be placed inside cutouts from other parts.

Rotation Step

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

Min Distance from Sheet

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

Plate Width Minimization

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

Placement Accuracy

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

Geometry Detail

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

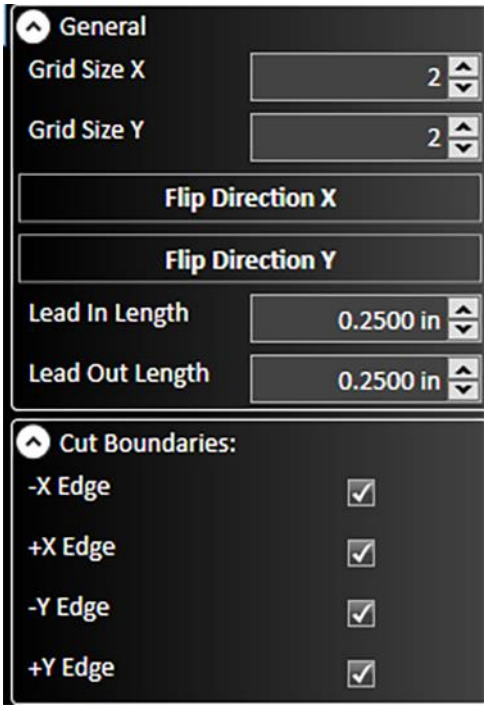
Maximum Contour Segment Count

Range 100-1000.

Common Line Cutting

Creates a set of identical parts based on a single part where the edges of each part are cut using a grid of common parallel lines. The part to be replicated must have parallel sides (e.g., rectangles).

Click the part to be replicated, and right-click to confirm. An outline view of the additional parts will be displayed. The parameters can be modified to change the number and orientation of the parts. After the changes have been accepted, the new toolpaths will be displayed.

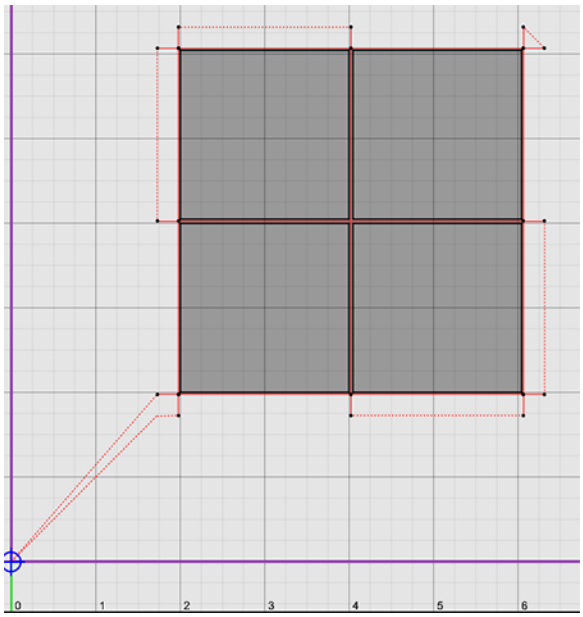


Grid Size X and Y

Number of parts created along either the X or Y axis.

Flip Direction X and Y

Changes the orientation of the new parts to the original part on the workpiece.



Lead In and Lead Out Length

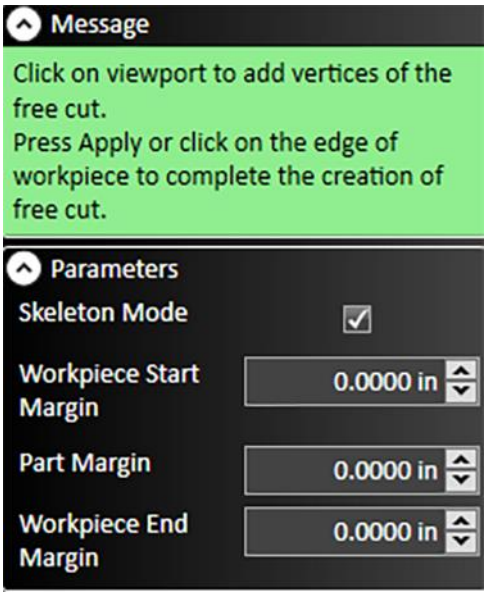
Sets the length of the cuts leading in or out from the perimeter of the grid.

Cut Boundaries

All boundary lines checked will be cut. To subtract a boundary, uncheck it. The + and - X and Y notations indicate the relative positions of the boundaries along each axis: in the example at right, the bottom, top, left, and right boundaries, respectively.

Free Path

Allows the user to add to the existing toolpaths on the drawing. Click to add vertices, similar to the Line tool in CAD.



Skeleton Mode

When selected, allows the user to define areas where cuts cannot be made, in relation to either the boundaries of the workpiece or any existing parts on the workpiece.

Workpiece Start/End Margin

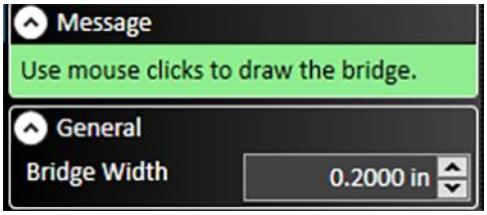
Defines the margin in relation to the leading or trailing ends of the workpiece.

Part Margin

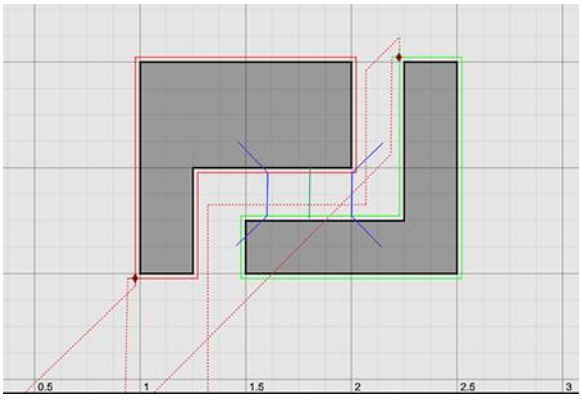
Defines the margin in relation to existing parts.

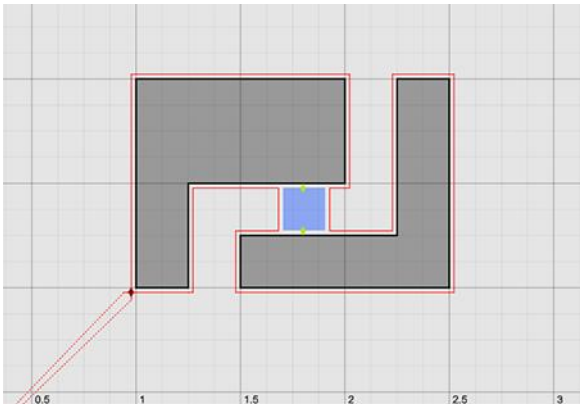
Bridge Objects

Allows the creation of a bridge between objects using the mouse. Click on the first object and then move the mouse to the desired endpoint. An outline of the bridge icon (below left) will be displayed. Click the second part to complete the bridge (below right).



The CAM project must be in **By-Route sequence mode**. See the **Connections Tool** for more details.

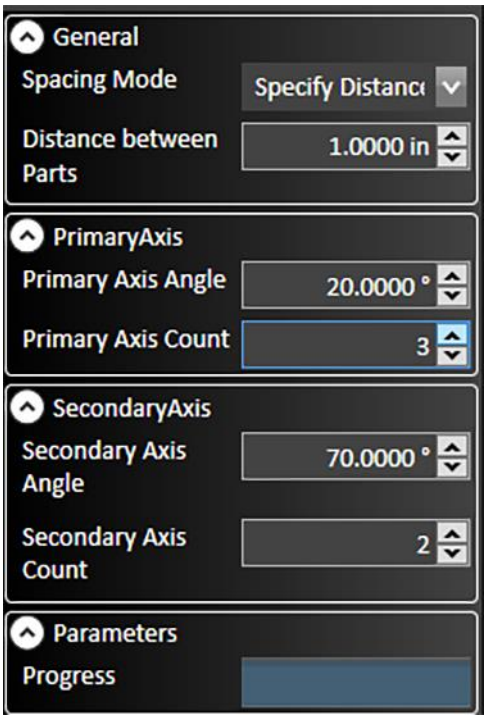




Layout (Pattern Nesting)

Creates a set of identical parts arranged in a pattern (a grid).

The number and orientation of parts in two axes can be set.



Spacing Mode

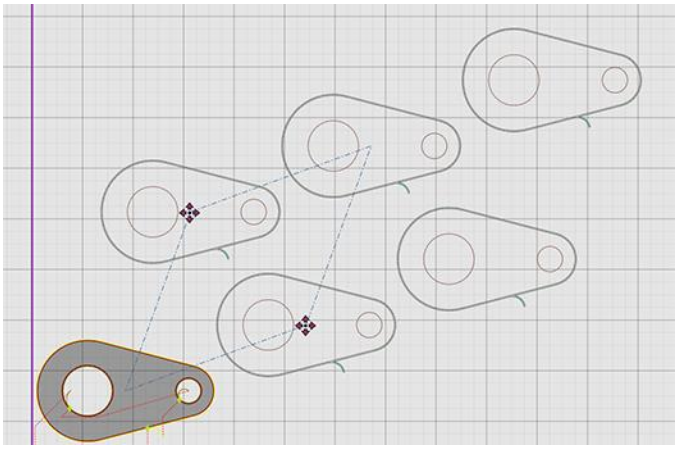
Specify offset or distance between parts.

Primary and Secondary Axis Angle

Set angle in degrees.

Primary and Secondary Axis Count

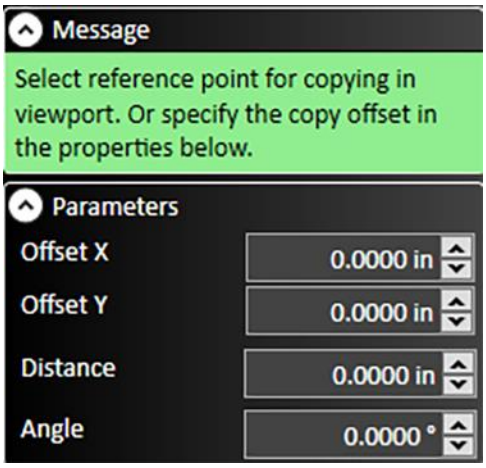
Number of parts to be generated along each axis.



Multi Copy

Allows copies of an object to be generated on the workpiece, either by positioning the cursor or by entering parameters.

Once an object has been selected and confirmed, a new copy can be placed with each cursor click.



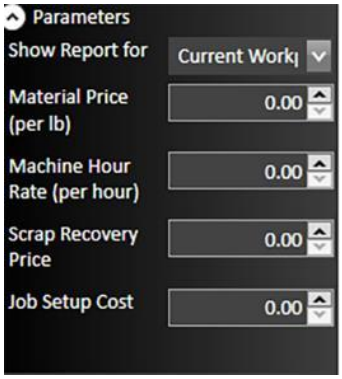
Nesting Information

Generates a report from the nested parts, estimating material use, waste, and cost. A single workpiece or all workpieces can be selected.

Figures for **Material Price**, **Machine Hour Rate**, **Scrap Recovery Price** and **Job Setup Cost** can be entered manually.

Other information is generated automatically based on the fab head configuration, material, and project settings.

The report can be saved (.txt), edited, and printed.



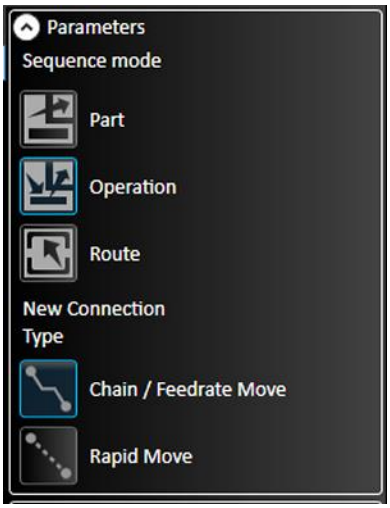
Fit workpiece to the parts

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

Connections tool

When selected, the connections 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.



Part sequence mode

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

Operation sequence mode

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

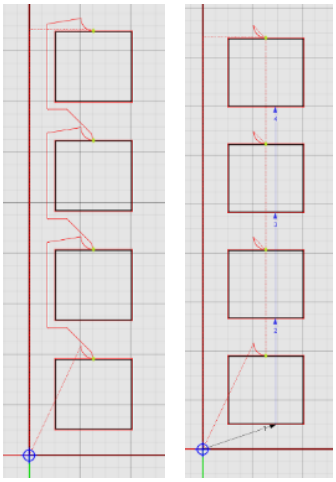
Route sequence mode

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.

Route sequence mode must be selected to enable chain cutting (example below).

Chain / Feedrate Move
Rapid Move

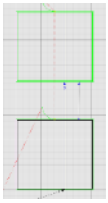


Chain cutting allows for cutting multiple parts without turning off the Fab Head. In the example at right, the same set of parts is shown with two different move types.

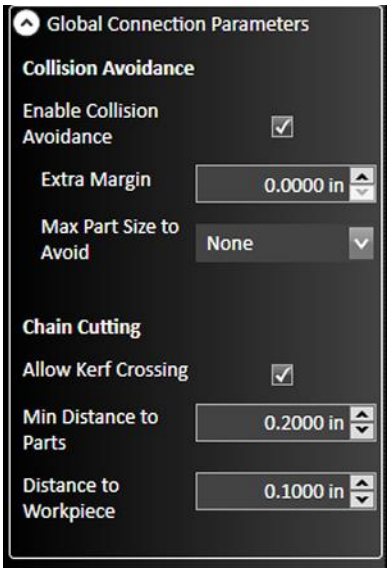
The first example (left) uses chain cutting. Note that the cutting path moves around the side of each part, leaving a margin.

The second example (right) uses rapid moves between each part. (This is the default).

The detail at right shows how to change the type of move between two parts:



1) Select the move type (chain/federate or rapid move).
2) Select the first part with the cursor.
3) Drag the cursor to the next part outline (highlighted in green).
4) Click and release the cursor.



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.

Extra Margin

Defines margin to add to path.

Max Part Size to Avoid

Define size by X dimension, Y dimension, or both X and Y.

Allow Kerf Crossing

When enabled, allows the Fab Head to pass back over existing cuts.

Min Distance to Parts

Define distance between chain cut and part.

Distance to Workpiece

Define distance between chain cut and edge of workpiece.

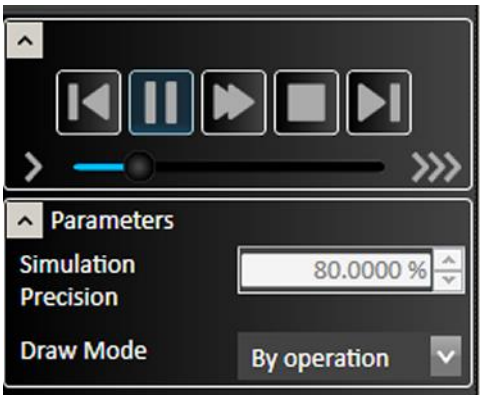
Simulation Tool

Allows visualization and analysis of the material removal process.

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

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

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

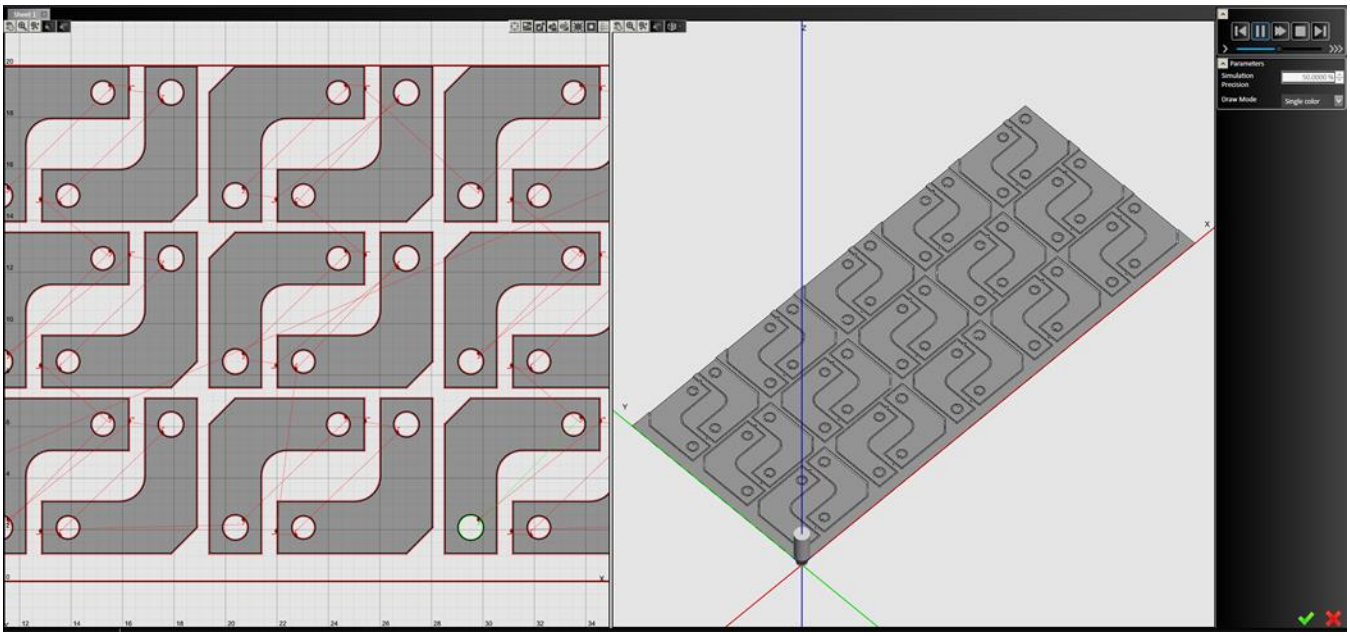


Simulation Precision

Range: 1-100%

Draw Mode

Single color, or by operation.



CAM Reset

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

Delete

Deletes the selected toolpath.

G-Code

The G-Code tools on the ribbon allow you to create or edit a G-Code program for the CAM process for the current workpiece. These are the tools that are available on the **G-Code** portion of the ribbon:

Create G-Code File

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

Open Editor

After a G-Code file has been created, you can open the G-Code editor window where you can view or edit the G-Code program

Import

Files can be imported into the CAM drawing workspace. If other objects are added, new toolpaths will be generated.

Note Objects imported as files will not appear in the CAD drawing space.

DXF/DWG

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

CADCAM

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

Viewport for CAM

In Flashcut CAM, the workspace is different for each type of drawing, such as 2D part drawing, 3D part drawing, and Pipe/Tube part drawing.

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.

Pan

Click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window allows you to pan without selecting the pan tool first.

Zoom

Click and drag anywhere to zoom in and out on the drawing. You can also use the scroll wheel to zoom; scroll up to zoom in and scroll down to move out.

Zoom Window

Click and drag a box in which to zoom in. This tool allows a user to focus on a specific area.

Zoom Workpiece Extents

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

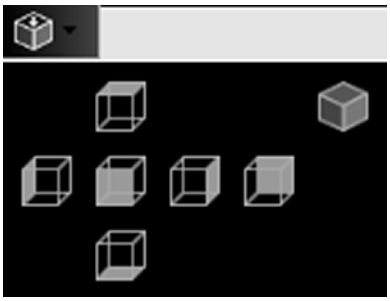
Zoom Toolpath Extents

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

3D Orientation Tool

The right-hand viewport 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.



Display Options

The display options tools are shown at the top of the drawing workspace and allow to 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.

Cutting Directions

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

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.

Material

When toggled, a red box is displayed in the drawing window 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.

Parts

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

Part Numbers

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

Rapid Moves

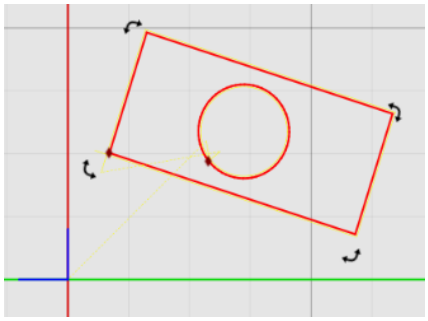
When toggled, rapid moves of the cutting tool are displayed on the drawing as dotted red lines. For rapid moves to be displayed, **Tool paths** must also be toggled.

Tool Paths

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

Manipulation Tools

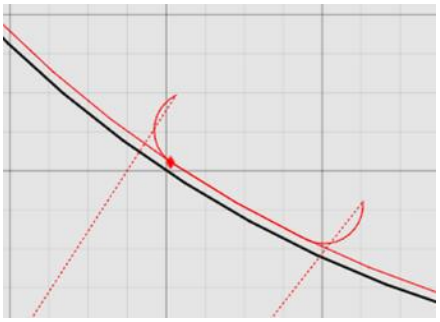
Feature Manipulation



You can maneuver parts in the drawing window into a desired rotation or position. Move the cursor over the part so that it 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.

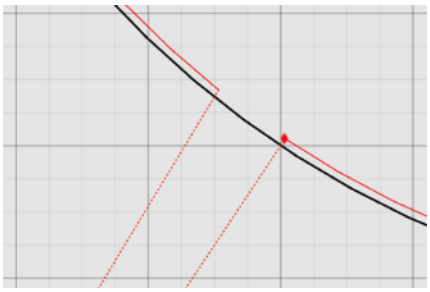
Lead In/Out Settings



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

The initial locations of these lines are automatically generated by an internal algorithm using basic rules. FlashCut CAM enables you to override these initial settings for each individual feature.

Editing Settings



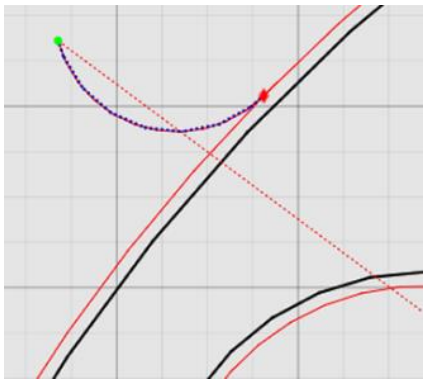
To adjust the lead in and lead out settings, select the red diamond icon on the desired feature. This brings up the parameters window.

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

In the examples to the right, the top drawing shows a closed break with an overburn. The cut begins with an arc-shaped lead in, and finishes with an arc-shaped lead out. The parameters for this break are shown below.

The bottom drawing shows a tab break, no lead in or lead out.

Manual Editing

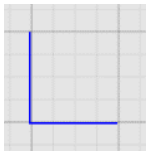


FlashCut CAM permits manual adjustment of individual lead ins and lead outs in the toolpath.

Click on the red diamond to make the line segment active. A blue dot appears at the end of the line. Hover over the blue dot until it turns green. This dot can now be used to drag the end of the lead in or lead out to the desired position.

The example at right shows a single lead in with its end point selected (cursor not shown for clarity).

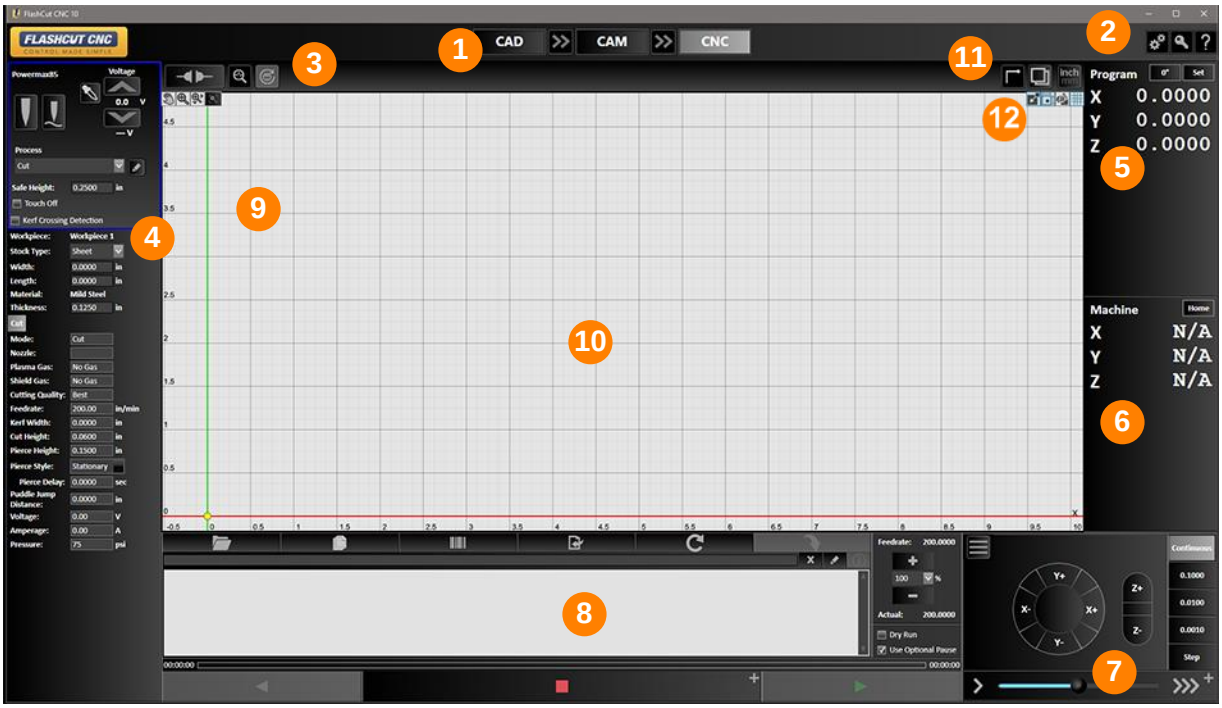
Program Zero



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.

FlashCut CNC

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



1. Navigating Through CAD CAM CNC
2. Configuration / License / Help
3. Hardware Controls & Diagnostics
4. Fabrication Head Controls
5. Program Coordinates
6. Machine Coordinates
7. Jog / Point Move / Rip Cut
8. G-Code Program Controls
9. Pan and Zoom Controls
10. Viewport for CNC
11. Workpiece Information
12. Display Options

Hardware Controls and Other Settings

Connect / Disconnect to Controller

Ensure that the signal generator (CNC controller) has the 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 the signal generator. If there is a problem with the connection, a dialog box appears.

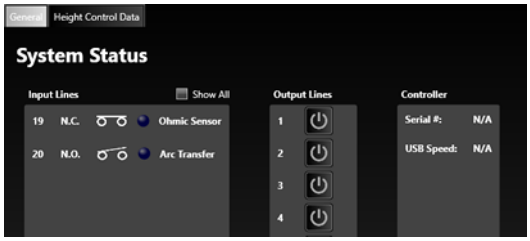
Before the unit connects, a safety reminder screen appears. It is imperative that you and anyone else near the machine understand, agree with, and adhere to all the safety guidelines. If the safety guidelines are not accepted, the software will not connect.

When properly connected, the **Connect** icon will join and turn green. Press **Disconnect** when you are using FlashCut CNC software and you do not want to control hardware.

System Status

Click the magnifying glass icon to view the status of the hardware configured on your system.

General



Input Lines

If a normally closed (NC) input line is **normal**, it will be in the **closed** state and the blue LED will **not** be illuminated. If the machine is connected, these icons are live when a switch changes its state.

If an NC switch is **tripped,** then it will be in the open state and the blue LED will be illuminated.

Normally, only input lines that have been defined are displayed. However, if the **show all** checkbox is selected, the status of all input lines will be displayed.

Output Lines

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

Controller

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

Height Control Data

When configured with a fabrication head that uses height control, diagnostic data from recent cuts can be viewed here.

Fabrication Head Diagnostics

When configured with a fabrication head with communications, a diagnostic data interface can be viewed here.

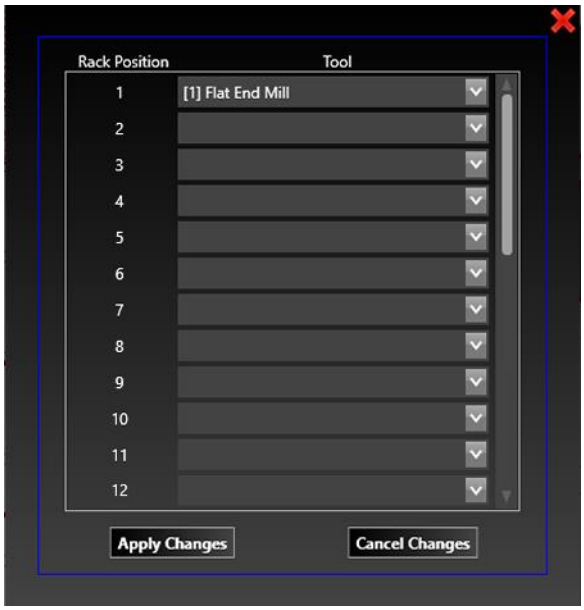
Reset Motor Drivers

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

Tool Rack Positions

When configured with a fabrication head that has automatic tool change, view and change tool rack positions here.

Each rack position displays a drop-down field with available tools. See **Tool Library** for more information.



Trace

Select how the toolpath in the CNC window is displayed during operations. Options are **Trace Off**, **Thin Trace**, and **Thick Trace**.

When **Thick Trace** is selected, the width of the trace includes the kerf.

Simulation

Clicking this icon toggles the simulation view for 3D operations.

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.

Workpiece Information

The stock type (Sheet, Block, Round Pipe, etc..), material, and workpiece dimensions are displayed here.

This information is retrieved from the loaded G-Code program. If there is no workpiece information within the G-Code program, you can use the Edit button to define the workpiece.

Fabrication Head Control

Each fabrication head that has been configured will be displayed here.

Each fabrication head will have a control section and a list of the process parameters will be displayed below the control section. When multiple fabrication heads are configured, process parameters for the currently selected fabrication head will be displayed.

All Fabrication Heads that are configured with a Lift Axis will have a Safe Height parameter.

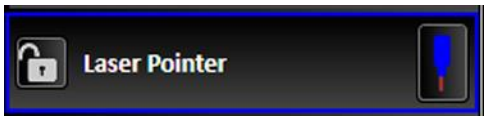
Safe Height

The height this fabrication head's lift axis moves up to during rapid moves.

Laser Pointer Controls

Laser Pointer On/Off

This button indicates when the Laser Pointer is On or Off (blue or gray). The Laser Pointer can be turned on or off manually with this button.



Lock Laser Pointer

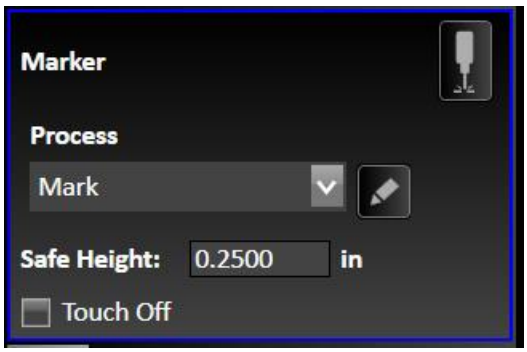
The Laser Pointer selection can be "locked". When it's locked, this allows you to do a type of 'dry run' of a G-Code program versus cutting or marking with the fabrication head specified in the G-Code program. If you want to have the "locked" Laser Pointer turn on and off while running a g-code program, similar to how the fabrication head would turn on and off, you'll need to set up the "Start/End of Point" macros in the Laser Pointer Fabrication Head Configuration. Be sure to "unlock" the Laser Pointer when you want to select or use a different fabrication head.

Marker Controls

The available controls depend on what features are configured for your marking system.

Marker On/Off

This button indicates when the Marker is On or Off (blue or gray). The Marker can be turned on or off manually with this button.



Process

Here you can manually select from the available processes. For example: Mark or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Marker Settings section in CAM.

Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Touch Off

Enable or disable the use of Touch Off.

Laser Controls

The available controls depend on what features are configured for your laser cutting system.

Laser On/Off

This button indicates when the Laser is On or Off (blue or gray). The Marker can be turned on or off manually with this button.

Process

Here you can manually select from the available processes. For example: Cut or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Laser Settings section in CAM.

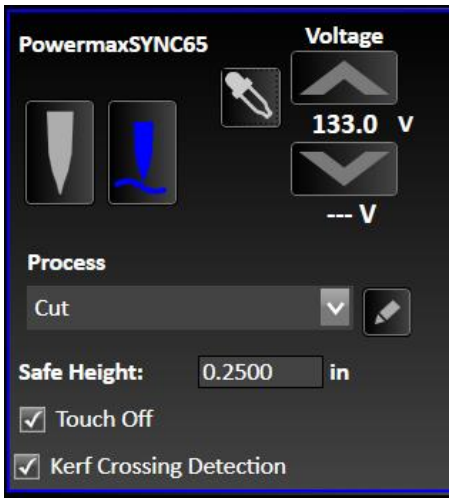
Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Plasma Controls

The available controls vary depending on the plasma system that is configured.

Torch On/Off

This button indicates when the plasma torch is On or Off (blue or gray). The plasma torch can be turned on or off manually with this button.



Torch Height Control

Enable or disable the use of automatic torch height control. Enabled is blue, not enabled is gray.

Sampling

Enable or disable the use of sampling for the set point voltage used by torch height control. Enabled is blue, not enabled is gray.

Set Point Voltage

This is the target voltage for torch height control. Also displayed is the actual voltage.

Process

Here you can manually select from the available processes. For example: Cut, Mark, Small Hole, Smart360, or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Plasma Settings section in CAM.

Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Touch Off

Enable or disable the use of Touch Off.

Kerf Crossing Detection

Enable or disable the use of kerf crossing detection to help avoid having the torch dive down into the material.

Oxyfuel Controls

The available controls vary depending on the oxyfuel system that is configured.

Preheat On/Off

This button indicates when the oxyfuel preheat is On or Off (blue or gray). If configured for bi-level preheat, there will be separate buttons for Low Preheat and High Prehate. The preheat can be turned on or off manually with the button.

Ignition On/Off

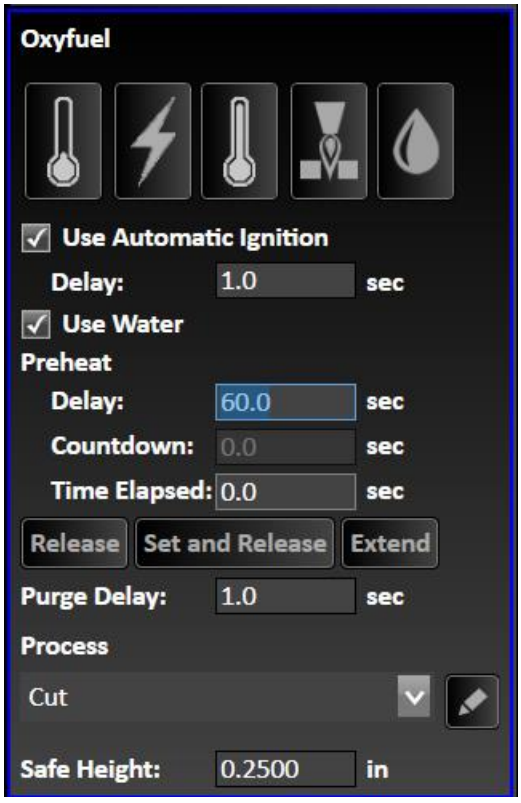
This button indicates when the oxyfuel ignition is On or Off (blue or gray). The ignition can be turned on or off manually with this button.

Oxyfuel Cut On/Off

This button indicates when the oxyfuel torch is On or Off (blue or gray). The oxyfuel torch can be turned on or off manually with this button.

Water On/Off

This button indicates when the oxyfuel water is On or Off (blue or gray). The water can be turned on or off manually with this button.



Automatic Ignition

Enable or disable the use of automatic ignition and set the corresponding delay.

Water

Enable or disable the use of water.

Preheat Controls

This displays the preheat delay time, along with a countdown timer and the elapsed time. Use the 'Release', 'Set and Release', or 'Extend' buttons to manually control the duration of the preheat process. The Release button will end preheating to proceed with cutting. The "Set and Release" button will end preheating to proceed with cutting, and a new preheat delay value will be set based on the elapsed time. The Extend button will keep the system doing a preheat indefinitely until a Release button is pressed.

Purge Delay

This is a delay time after turning off the oxyfuel before proceeding.

Process

Here you can manually select from the available processes. For example: Cut or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Oxyfuel Settings section in CAM.

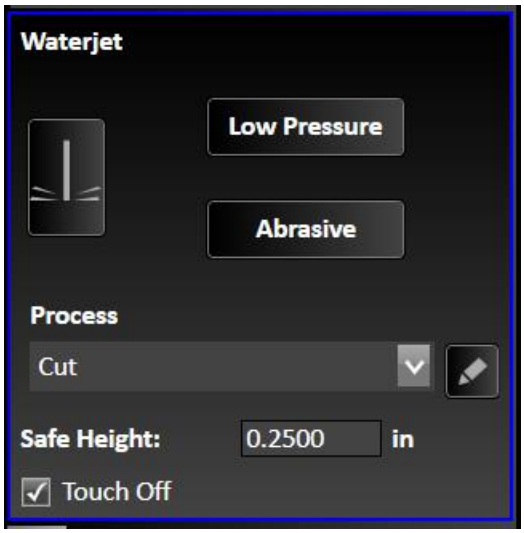
Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Waterjet Controls

The available controls vary depending on how the waterjet system is configured.

Waterjet On/Off

This button indicates when the waterjet is On or Off (blue or gray). The waterjet can be turned on or off manually with this button.



Low Pressure On/Off

This button indicates when Low Pressure is enabled. (blue is enabled) The waterjet can be turned on or off manually with this button.

Abrasive On/Off

This button indicates when the use of Abrasive is enabled. (blue is enabled) The abrasive can be turned on or off manually with this button.

Process

Here you can manually select from the available processes. For example: Cut or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Waterjet Settings section in CAM.

Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Touch Off

Enable or disable the use of Touch Off.

Spindle Controls

The available controls vary depending on how the spindle is configured.

Spindle On/Off

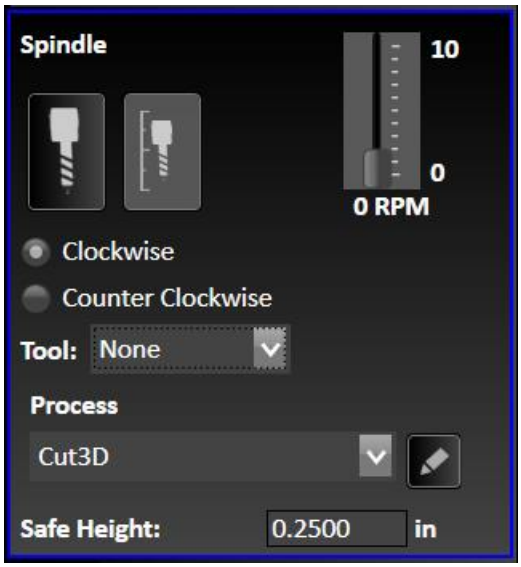
This button indicates when the spindle is On or Off (blue or gray). The spindle can be turned on or off manually with this button.

Clockwise / Counter Clockwise

If configured for bi-directional spindle control, these buttons indicate the direction of the spindle.

Spindle Speed

If configured for spindle speed control, there will be an indicator for the spindle speed. The speed can be controlled manually with the slider.



Tool Length Sensing

Use this button to manually initiate tool length sensing of the currently selected Tool. If no Tool is loaded, you will be given the option to sense the chuck. Machine coordinates must be set to use this feature.

Tool Menu

This displays the currently loaded tool. You can manually select a different tool from the menu. Note: If you are configured to use an automatic tool changer, selecting a different tool will initiate an automatic tool change.

Process

Here you can manually select from the available processes. For example: Cut3D or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Spindle Settings section in CAM.

Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Hotwire Controls

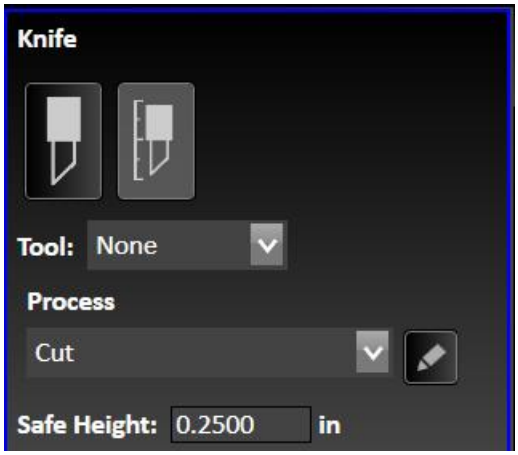
When a hotwire is configured, it can be controlled in CNC. Note: CAM does not support toolpath generation for hotwire.

Tangential Knife Controls

The available controls vary depending on how the knife is configured.

Knife Oscillation On/Off

This button indicates when the knife oscillation is On or Off (blue or gray). The oscillation can be turned on or off manually with this button.



Tool Length Sensing

Use this button to manually initiate tool length sensing of the currently selected Tool. If no Tool is loaded, you will be given the option to sense the chuck. Machine coordinates must be set to use this feature.

Tool Menu

This displays the currently loaded tool. You can manually select a different tool from the menu.

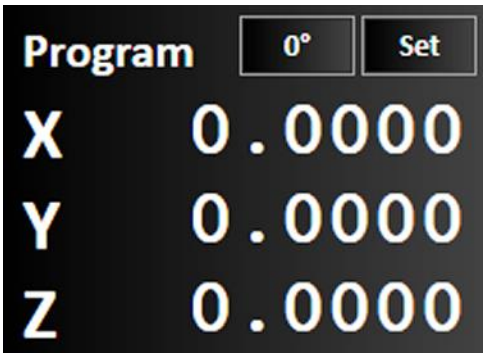
Process

Here you can manually select from the available processes. For example: Cut, Mark, or a custom Process that is defined within the loaded G-Code program.

View the Process parameter settings in the area below the fabrication head controls. For more information about each of the Process parameter settings, review the Knife Settings section in CAM.

Open the Process Editor dialog to temporarily change some process values. Edited values will become highlighted.

Program Coordinates Panel



The program coordinates refer to the exact position of the tool with respect to **Program Zero**.

The program coordinate system is referenced by the G-Code file as the set of absolute coordinates. Axes in the workspace represent the program coordinates.

The panel has two dropdown menus:

Coordinate Rotation (**0°**)

Set

The **Coordinate Rotation** dropdown menu (**0°**) 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 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 / Move to Point** / Rip Cut Control Panel) and then selecting an item from the dropdown menu. Options include:

Lower Left and Lower Right

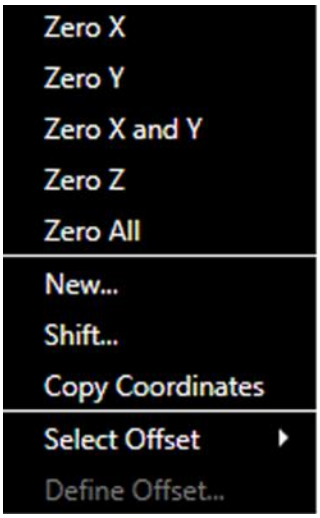
Lower Left and Upper Left

Lower Right and Upper Right

Upper Left and Upper Right

Enter Rotation Angle allows manual entry of a value in degrees. The angle may be positive or negative.

Clear Rotation resets the angle of rotation to zero degrees.



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.

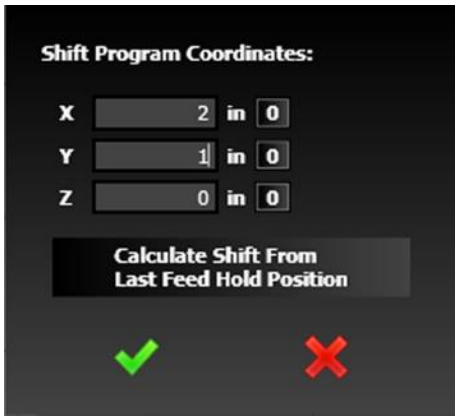
New... opens a dialog where you can set the position of the tool relative to program zero.

Shift... opens a dialog where you can set the position of the tool relative to the last feed hold position

Copy Coordinates copies the coordinates of all axes to the clipboard as text

Select Offset

Define Offset



Coordinate Rotation Example

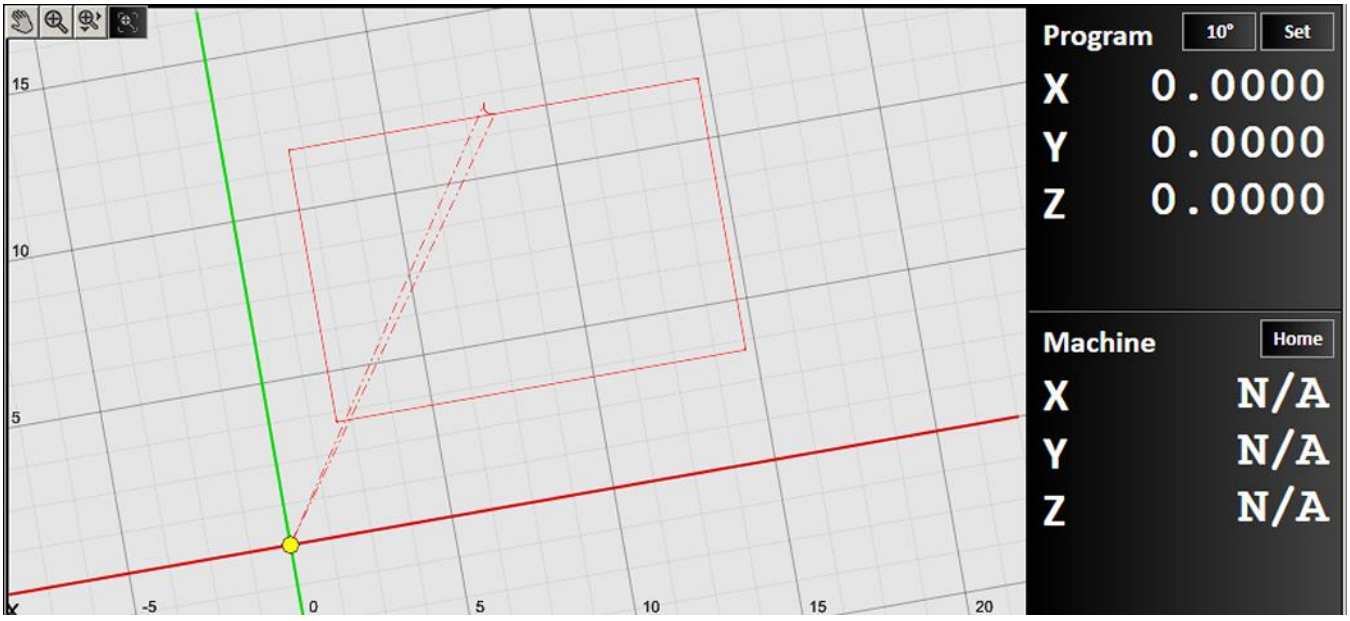
Coordinates rotation can be used to accommodate a workpiece that is too heavy or inconvenient to move on the machine. In effect, the program axes can be adjusted to conform to the dimensions and orientation of the workpiece.

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.

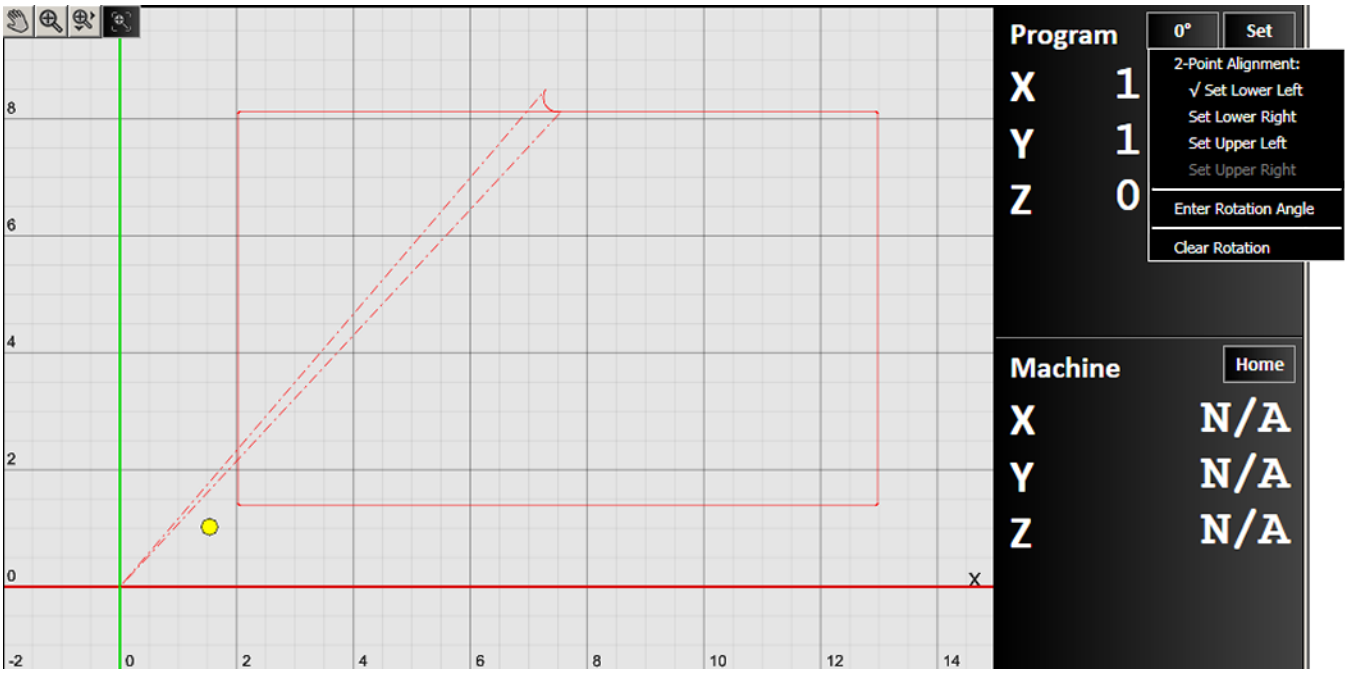


Use 2-Point Alignment

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

3. Jog the tool position to the lower right corner of the workpiece.

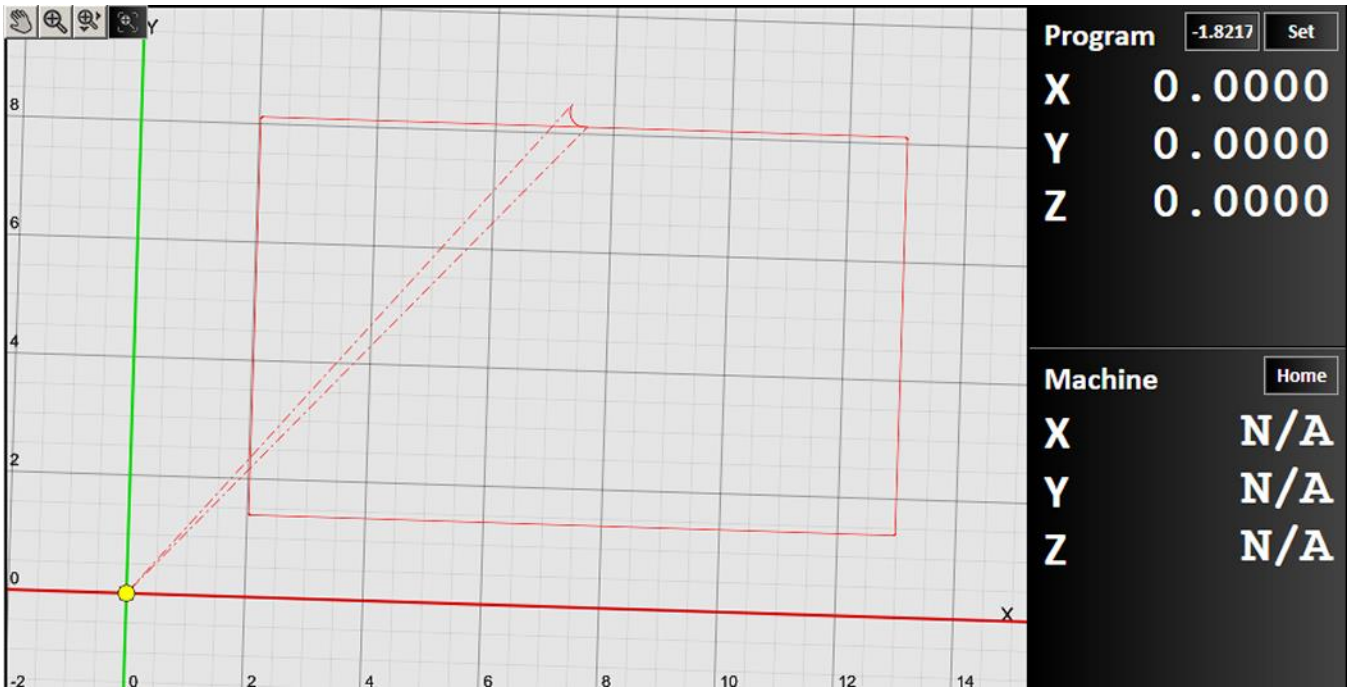
4. Select **Set Lower Left** from the dropdown menu.



5. Jog the tool position to the lower right corner of the workpiece.

6. 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.

7. 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.

Machine Coordinates Panel

The machine coordinates refer to the exact position of the tool with respect to the machine home. The coordinates will read **N/A** until the tool is able to locate its home position.

After connecting the signal generator to FlashCut, it is necessary to seek out and set the home point, or **Machine Zero**, of the tool. It is recommended that each axis is jogged first near the home switch before homing. Once machine zero is set, the machine tool envelope is redefined in the workspace.

The **Home** dropdown menu provides the following commands:

Home All will reset all three axes to the home point.

Zero All will set the current point as the origin of the machine coordinates.

Clear will close the machine coordinate system. **N/A** will display on each axis. You will need to reestablish a machine zero point.

Copy Coordinates copies the coordinates of all axes to the clipboard as text

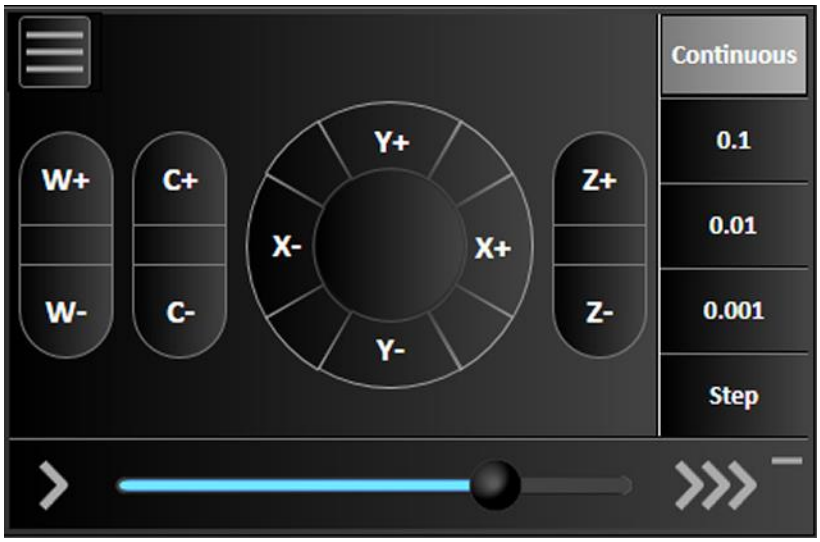
When multiple motors are used to drive an axis, FlashCut simultaneously homes each motor independently. For example, if two motors are driving a single axis, FlashCut starts the homing process by moving both motors towards the homing limit switches. Once one motor reaches its limit switch, it stops and 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 actuators.

Jog / Move to Point / Rip Cut Control Panel

The 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.

Jog

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



Axis Jog Buttons

Pressing an axis jog button moves the machine tool along the selected axis in the chosen direction. Ramping is used in cases where the jog rate is faster than the stop/start feedrate.

For the X and Y axes, the buttons are arranged in a circle. Pressing one of the diagonal jog buttons moves the machine tool along both axes.

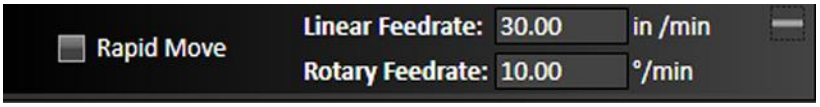
Jog Mode

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

Jog Rate

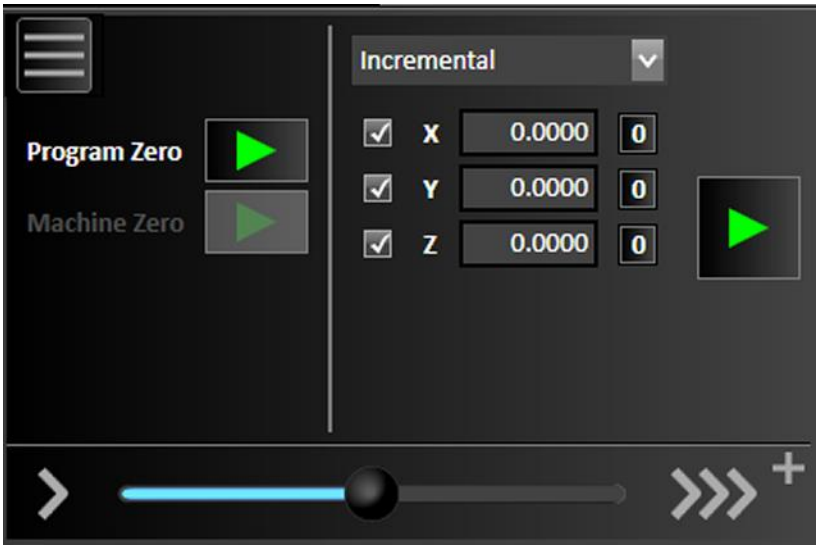
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

If you want to move at the maximum feedrate or at a specific feedrate, toggle the **Plus icon** to set those feedrate options.



Move to Point

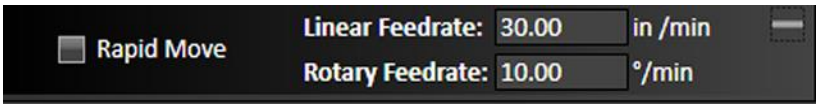
Move-to-point allows you to move (jog) your tool to a specified exact point. You may specify the feed rate to be used for these moves by editing the value in the **Feedrate** field.



Click the **Program Zero** button to move the tool to the program zero point shown on the **Program Coordinates Panel**.

Click the **Machine Zero** button to move the tool to the machine zero point shown on the **Machine Coordinates Panel.**

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



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. The choice you make with the dropdown 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, two, or all three 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 "zero" **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.

Move Rate

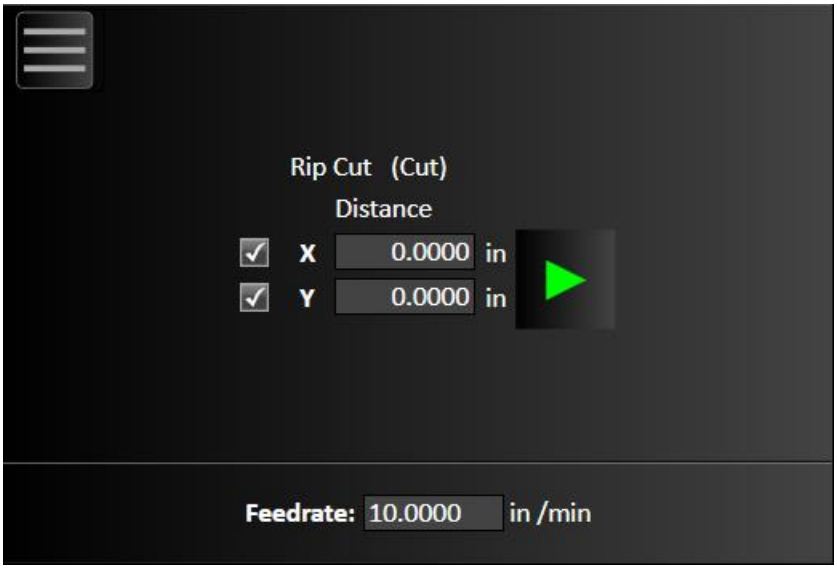
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

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.



G-Code Control

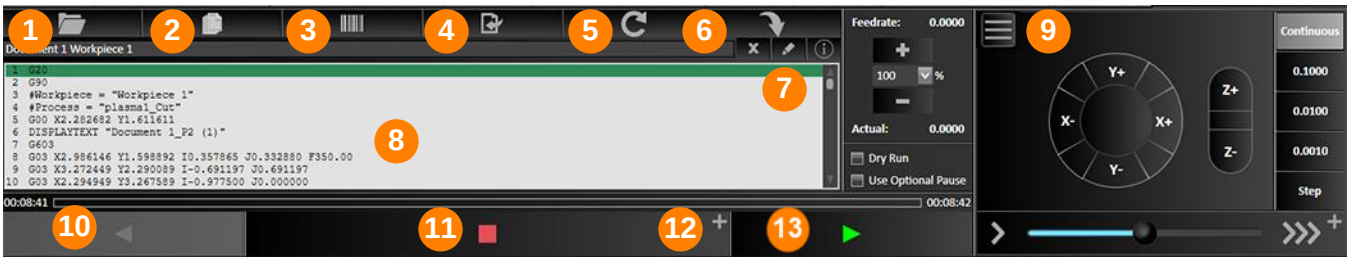
FlashCut supports ANSI standard G-Code to control machine tool movement and peripheral devices. This section describes how to create, open, and modify G-Code files, and the G-Codes supported. The G-Code window provides numerous options for creating, opening, editing, and running G-Code files. There are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

Create a G-Code file through FlashCut CAD and FlashCut CAM. Clicking the scissors button in either of these modes generates lines of G-Code corresponding to the design specifications.

Open an existing G-Code file created by FlashCut, another CAM program, or any other source.

Write a G-Code program directly in the FlashCut G-Code editor.

Double-click the G-Code window to launch the G-Code editor. The active file will be loaded.



1. Open G-Code File
2. Load Multiple Files
3. Barcode
4. Run CAD Import Wizard
8. G-Code
9. Feedrate Override
10. Run G-Code in Reverse
11. Feed Hold

5 Reset G-Code

6 Jump To Line

7 Close G-Code File
Edit G-Code

12 Toggle G-Code Run Mode

13 Run G-Code

Open G-Code file

When a CAD or CAM design is sent to be manufactured in FlashCut CNC, it appears in the G-Code. 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. A visual preview of the code will appear in the workspace.

This workspace displays the currently loaded G-Code file. While the program is running the current line of code is highlighted in real time, this way the user can track their progress throughout the operation.

File name

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

Load Multiple G-Code Files

Prompts the user to select multiple G-Code files. All selected G-Code files will be loaded to run.

Barcode

Allows the user to scan a barcode that corresponds to a G-Code file to load.

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.

Reset G-Code

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

Jump to Line

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

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.

Edit G-Code

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

G-Code Program Listing Box

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.

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.

Run G-Code in Reverse

Runs the program in reverse. This is useful when a problem occurred while cutting and you need to move the fabrication head back to the desired position to resume cutting.

Feed Hold

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

Toggle G-Code Run Modes

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.

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.

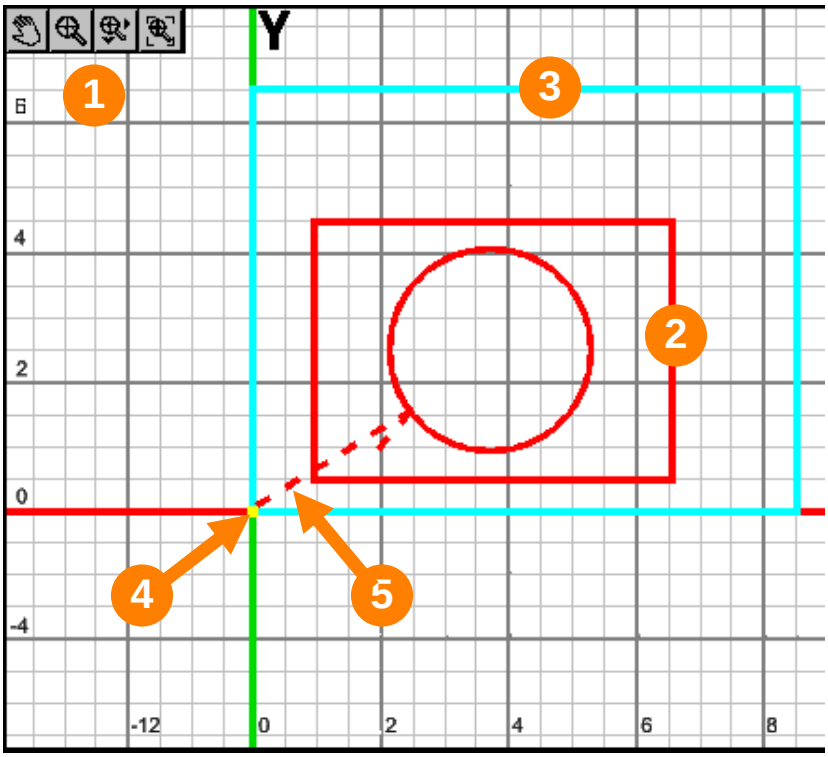
Progress meter

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.**

Viewport for CNC

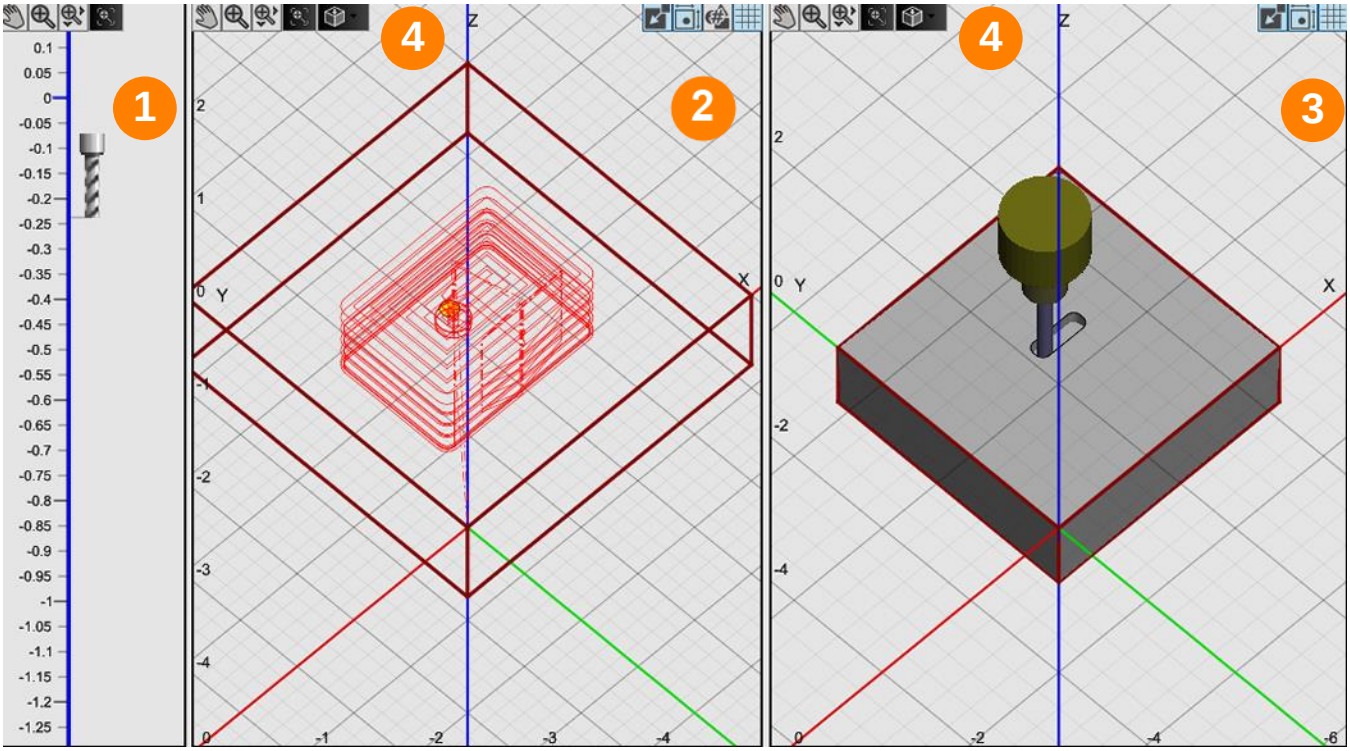
The workspace displays different views of the tool path described by the current G-Code file.

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. Red solid red lines show the toolpath preview.
3. Machine Envelope
4. The current tool position is shown with a yellow dot (or crosshairs).
5. Red dashed lines show a preview of rapid moves.

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 3D Orientation Tool

Pan and Zoom Tools

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

Pan

Click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window allows you to pan without selecting the pan tool first.

Zoom

Click and drag anywhere to zoom in and out on the drawing. You can also use the scroll wheel to zoom; scroll up to zoom in and scroll down to move out.

Zoom Window

Click and drag a box in which to zoom in. This tool allows a user to focus on a specific area.

Zoom Extents

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

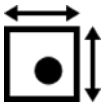
Display Options

Material

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

Machine Envelope

When selected, displays the machine envelope. The machine coordinates must be set/homed and the controller must be connected for this option to be available.

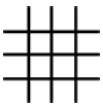


Part Numbers

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

Show/Hide Grid

Toggles the **grid** in the active drawing workspace.



Help

Selecting the Help icon from 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 in on a support call.

System Info

This displays the software version, the serial number of the controller (if the controller is Connected), the firmware version of the signal generator.

Support Contact

This displays the company name, website, email, and phone number to use for support. There are also links to websites that have instructional videos and other information for several features of the software.

User's Guide

This will open the User's Guide (this document) that is included in the software.

Programming Reference

This will open the Programming Reference document that is included in the software. The Programming Reference contains information about G&M-Codes, program variables, and other programming information for advanced users.

Build Support File

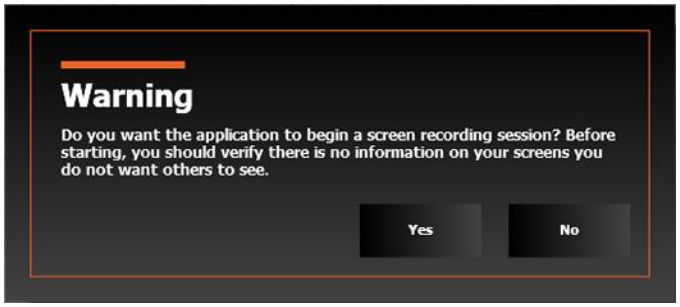
A support file is a compressed ZIP file that contains all current information about your system, including copies of current configuration, open CAD/CAM files, the G-Code file, and log files. It will also include information from the Steps Recorder, if used (see below). This information can help FlashCut technical support answer a question or diagnose a problem.

FlashCut displays the file **Save As** dialog so you can save the support file in a convenient location for attachment to an email message. The default location for the saved file is **C:\FlashCut Data\Support**.

Steps Recorder

If you are encountering a problem or have questions about a particular procedure, you can utilize the Steps Recorder Feature. When you use this feature, a screen recording session will begin. Go through the steps that you want technical support to see, then press the Stop Recording button. Then build a support file. This Steps Recorder session will be included within the support file.

The following warning dialog is displayed:



While a screen recording session is in progress, a red square icon will appear in the upper right of the program window. Press the **Stop Recording** button when you're done recording.

Software License Agreement

Displays the text of the Software License Agreement.