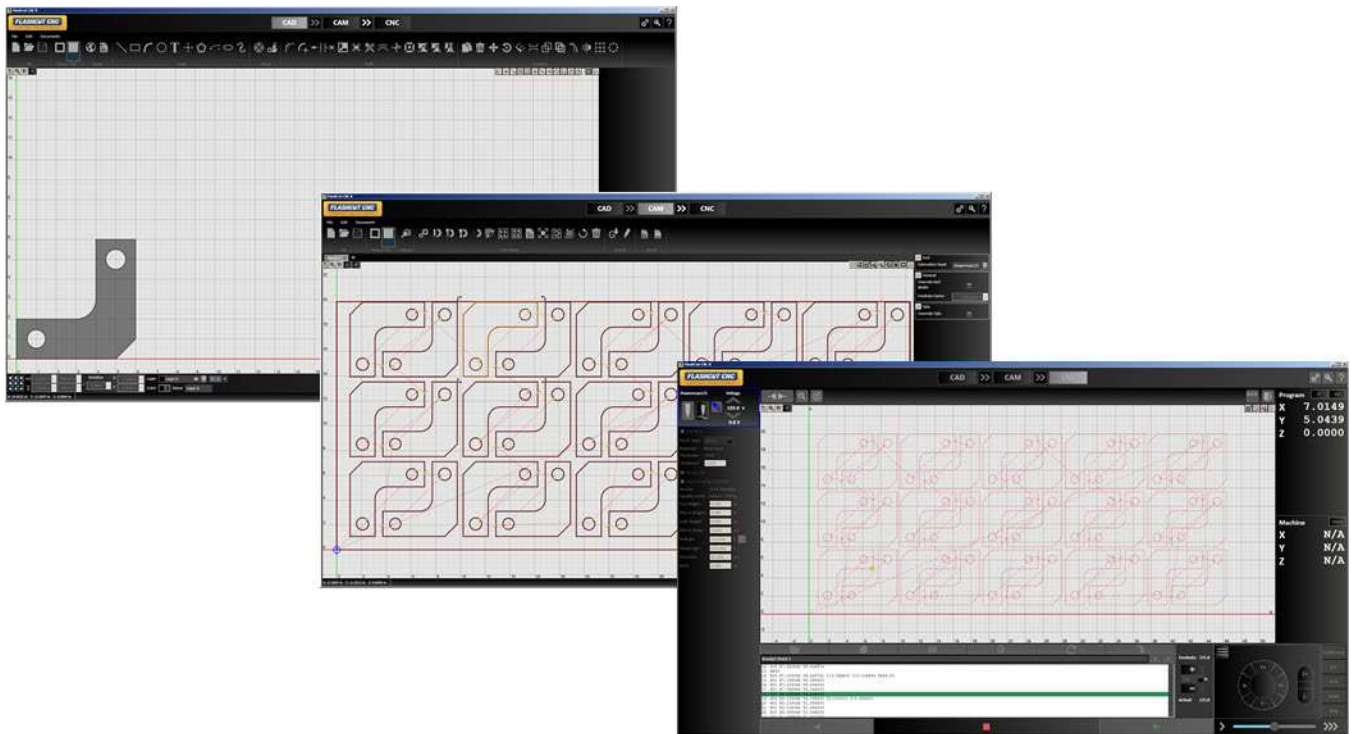


FLASHCUT CNC

CONTROL MADE SIMPLE



FlashCut CAD/CAM and CNC Control Software



Notices

Revised July 9, 2020

Version 9.0.0.103

© 1997-2020 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 of 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)
- CAD/CAM 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.

Note The Support file will only include a G-Code file that has been saved and it **will not** include a CAD/CAM file. If you are having a CAD/CAM issue, please e-mail the CAD/CAM file separately.

Please see **Getting Help** 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

Contents

Notices	ii
Introducing FlashCut® CNC.....	11
System requirements	11
Program packages and features	11
Version comparison	11
Installing the FlashCut CNC software	14
Preparing to install the software	14
Installing the software.....	14
Updating the setup file	18
Installing the USB driver	19
Windows 8 and 10	19
Windows 7	19
Getting started with FlashCut CNC.....	24
Launching the software	24
Running in evaluation mode	24
Activating a license	24
Configuring the software	24
Getting help.....	25
Understanding common interface elements	25
Configuration, License, and Help buttons	25
Tabs.....	25
Accept / Cancel.....	26
Menu bar	26
Documents menu	32
Part shading and Grid tools.....	32
Parameters area	32
Status bar.....	32
FlashCut CAD	33
Ribbon.....	34
File tools	34
Feature Type	35
Import tools.....	35
Create tools	46
Shapes	58
Modify tools.....	60
Group	68
Ungroup	68
Ungroup All	68
Transform tools.....	69
Pan and zoom tools	79
Pan	79
Zoom Extent	79
Zoom.....	79
Zoom to Fit	79
Snap tools	79

All snap	80
Center snap	80
Endpoint snap	80
Grid snap	80
Horizontal/vertical snap	81
Intersection snap	81
Midpoint snap	81
Nearest snap	82
Parallel snap	82
Perpendicular snap	82
Quadrant snap	83
Tangent snap	83
Free form stretching and manipulation tools	83
Control handle grid	84
Horizontal and vertical dimensions	84
Rotation	84
X and Y position	85
Layers	85
Layer	86
Color	86
Name	86
Drawing workspace	86
Viewport tools	88

FlashCut CAM 89

Menu bar print commands	90
Print	90
Ribbon	93
File tools	93
Feature Type	93
Selection	94
CAM Actions	95
Import	120
DXF/DWG	120
CAD/CAM	120
Pan and zoom tools	121
Pan	121
Zoom Workpiece Extents	121
Zoom	121
Zoom Toolpath Extents	121
Zoom Window	121
Orientation tool	121
Display options tools	122
Cutting Directions	122
Kerf	122
Material	122
Parts	122
Part Numbers	122
Rapid Moves	122
Tool Paths	123
Drawing workspace	123

Feature manipulation	123
Lead in/out settings	123
Program zero	124
FlashCut CNC	125
System Status, Connect, Reset Motor Drivers and Tool Rack Positions	126
System Status	126
Connect	126
Reset Motor Drivers	127
Tool Rack Positions	127
Fab head settings	128
Laser pointer control	128
Marker control	128
Laser cutter settings	129
Plasma torch settings	130
Oxyfuel settings	131
Waterjet settings	132
Spindle settings	133
Hotwire Settings	133
Tangential Knife Settings	134
Program coordinates panel	135
Coordinate rotation example	136
Enter rotation angle	136
Use 2-point alignment	137
Machine Coordinates Panel	139
Jog and point control panel	139
Jog	140
Move to point	141
Rip Cut	142
G-Code window	143
Pan and zoom tools	146
Drawing workspace	147
Multiple views for 3D operations	148
Toggle Display Units	149
Toggle manual control	149
Material	149
Machine Envelope	149
Part Numbers	149
Show/Hide Grid	149
Configuring FlashCut	150
Menu selection tab	152
Setup Tab	152
Tooling Tab	152
Setup file (*.setup extension)	152
Tooling file (extension *.tooling)	153
Configuration workspace	153
Setup Configuration Menu	154
System	155
General	156
Controller	156

Safety	160
Preferences.....	161
Language.....	161
Jogging.....	161
Display	163
Pendant.....	164
USB Interface Board.....	166
Machine	167
Fabrication Heads	167
Fabrication Head Change.....	192
Axes	193
Drive Parameters.....	200
Rotary Axes	205
Reference Points	207
Fixture Offsets	208
Homing	209
Dynamic Ventilation.....	212
Dynamic Ventilation Example.....	213
Tool Changer	214
Sensing	216
Tool Length Sensing.....	216
Programming	217
M-Code Definitions	217
M-Code Execution	219
G-Code.....	220
I/O	221
Input Lines	221
Output Lines	224
User Interface	225
Custom Buttons.....	225
CAD	226
General.....	226
Features	226
CAM	227
General.....	227
Features	227
Post Processor	228
Tooling Configuration Tab.....	231
Tool Library.....	231
Licensing FlashCut	235
License Management Window	235
Import License File.....	235
Activate PC License.....	236
Deactivate PC License	236
Licensed Features List	237
Getting Help	238
Help.....	238
Version.....	238
Firmware Version	238

Support Contact238

User’s Guide238

Programming Reference238

Build Support File.....239

Steps Recorder239

Introducing FlashCut® CNC

Thank you for purchasing FlashCut® CNC, the most innovative, powerful CNC control system for Windows 7, 8, and 10. 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 Version 9 has three main portions – FlashCut CAD, FlashCut CAM, and FlashCut CNC.

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

FlashCut CAD and FlashCut CAM are sold as a combined product. FlashCut CNC is sold separately, but all three 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 Version 7, 8, or 10 (64 Bit)
- At least 4GB of RAM
- Intel I5 processor (or equivalent), or better
- A separate graphics card is preferred for processing larger files

Program packages and features

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

Version comparison

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

Version 9.0 Feature Summary	Standard Package	Pro Package	Premium Package
Simple, powerful 2-D CAD with improved DXF/DWG import	x	x	x
Measuring tool displays any part dimension or distance	x	x	x
Free form stretching and manipulation	x	x	x
Layer support	x	x	x
Centerline image import	x	x	x
Silhouette image import	x	x	x
Integrated CAM with true shape nesting with improved performance	x	x	x
Commonly used plate list	x	x	x
Customizable cut charts	x	x	x
Customizable default lead-in, lead-out strategies	x	x	x
Editable plasma and oxyfuel cut charts for Hypertherm® and Thermal Dynamics®	x	x	x
Oxyfuel support	x	x	x
Plasma support	x	x	x
Automatic kerf compensation	x	x	x
Grid nesting	x	x	x
Graphical editing of lead in and lead out locations	x	x	x
Tabbing	x	x	x
Overburn support	x	x	x
Assign different operations to each part	x	x	x
Cut sequencing	x	x	x
Drag, Delete, Rotate, Copy parts on the nest	x	x	x
Operation sequencing	x	x	x
Cutout or Part Mode	x	x	x
Custom views of kerf width, cut direction, toolpath, rapids and part geometry	x	x	x
Customizable post processor	x	x	x
Easy CNC program flow control including jump to line, forward, and reverse processing	x	x	x
Dry run CNC	x	x	x
Momentary run CNC	x	x	x
Kerf crossing detection	x	x	x
THC performance history chart	x	x	x
Smart feed rate and Torch Height Control (THC) processing for small holes	o	x	x
Curved text around any shape	o	x	x
DXF export	o	x	x
Marker and plasma marking	o	x	x
True type font support without exploding	o	x	x

Mill router	o	o	x
3D cutting simulation	o	o	x
Laser pointer and laser pointer software support*	o	o	o
Integrated shape library	o	x	x
Fix drawing tool finds and corrects CAD mistakes	o	x	x
Bridge entities tool	o	x	x
Shape welding tool	o	x	x
Part shading	o	x	x
True shape nesting with improved performance	o	x	x
Import of parts from external files	o	x	x
Multi-sheet and fill sheet true shape nesting	o	x	x
Corner looping	o	x	x
Automatic Lead-In on Restart	o	x	x
Dynamic THC on/off	o	x	x
Rip cutting	o	x	x
Advanced G-code file management	o	o	x
Dynamic ventilation	o	o	x
Coordinate rotation for workpiece alignment	o	o	x
Rotary axis support	o	o	o
Compound (dual) rotary table support	o	o	o
Programmable I/O interface board*	o	o	o
Joystick with Programmable I/O Interface Board *	o	o	o
Integrated THC with Mini I/O Expansion Board, Voltage Isolation Box, Ohmic and Torch Interface Cables *	o	o	o
RS-485 communications with Hypertherm® Powermax® plasmas for real-time current and pressure control *	o	o	o
Smart touch off	o	o	o
Laser cutter support		o	x
Support for multiple simultaneous fabrication heads		o	x
Part numbers		o	x
Waterjet support		o	x
Multi-layer color image import		x	x
Advanced lead in and lead out editing		x	x
Animated 3D cutting simulation		x	x
Nesting report to estimate cost and weight of materials		o	x
CNC progress meter		x	x
Move to point in CNC viewport		x	x
Motor current sensing		o	o
* Includes hardware			

Installing the FlashCut CNC software

Preparing to install the software

Important: Install FlashCut CNC before connecting to a signal generator

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

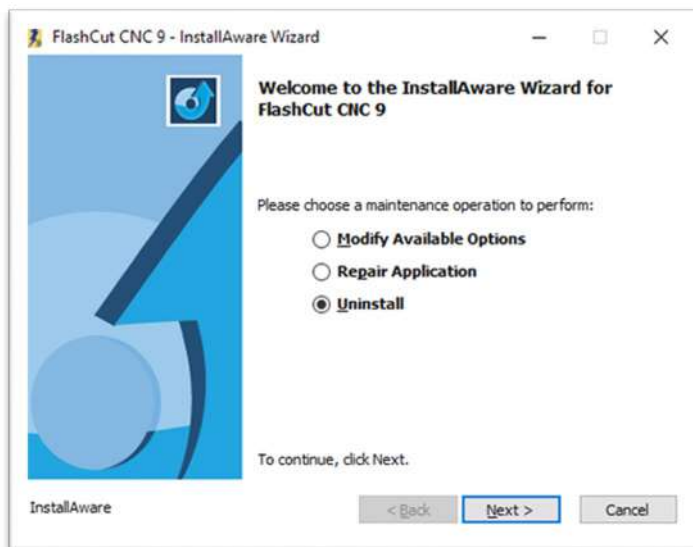
When the hardware is connected to the computer with the USB cable and then powered on, Windows will try and find a driver for the device. If the correct driver is not already installed, Windows may attempt to use a driver for a different device. If the software is installed first, the proper driver will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

Installing the software

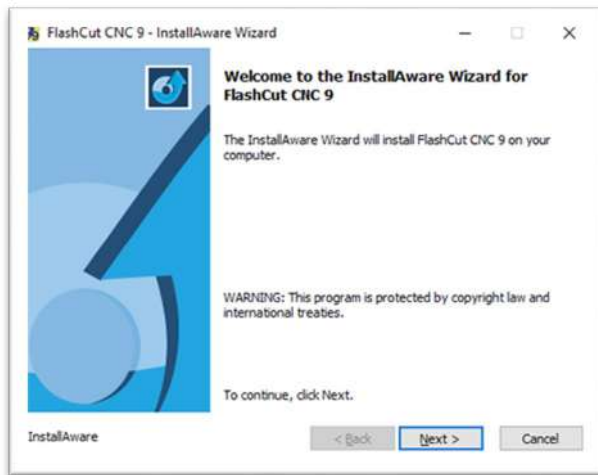
Please follow these instructions to install FlashCut CNC.

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

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

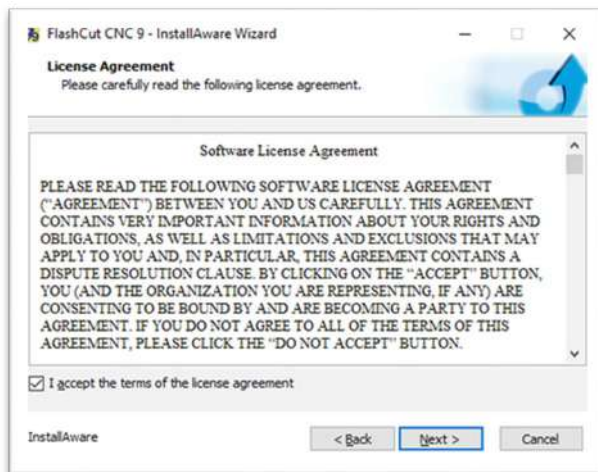


4. After dealing with the pre-installation requirements, the wizard prompts you to install the latest release.



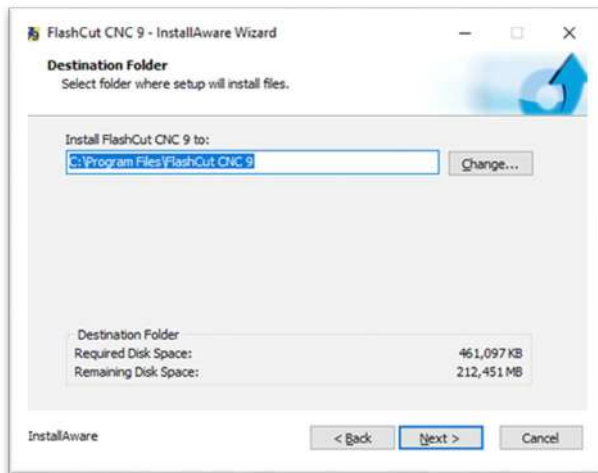
Click **Next >**.

5. Review the software license agreement.



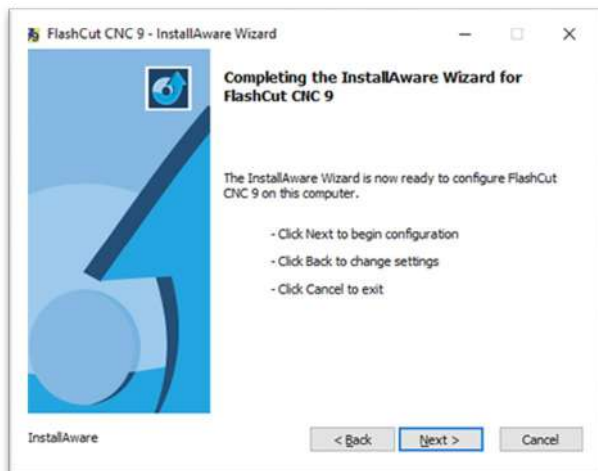
Click **Next >**.

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



Click **Next >**.

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



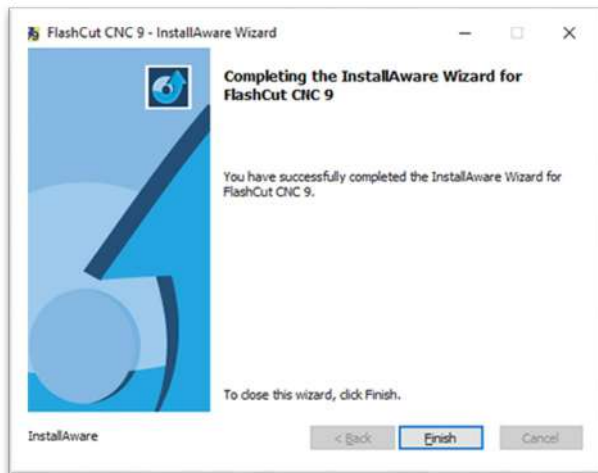
Click **Next >** to finalize the configuration and begin the installation.

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



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

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

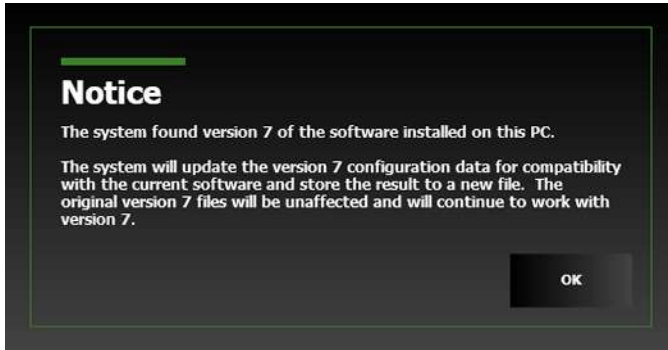


Click **Finish** to close the wizard.

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

Updating the setup file

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



For more information on the setup file, see [Configuring FlashCut](#).

Installing the USB driver

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

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

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

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

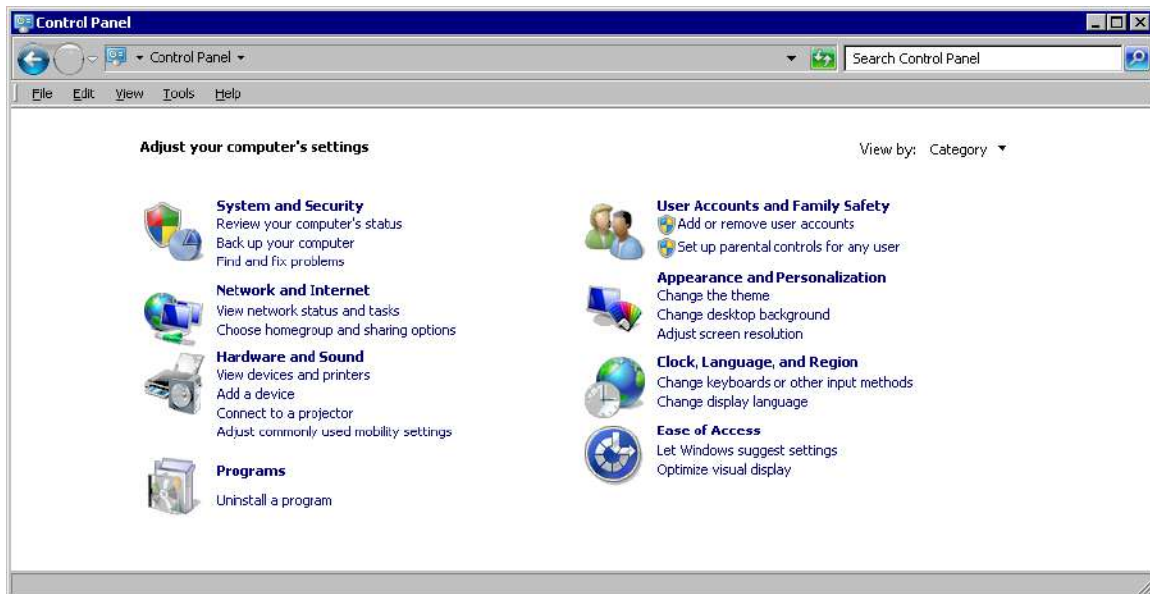
Windows 8 and 10

Later Windows versions automatically find and install the driver for the USB signal generator.

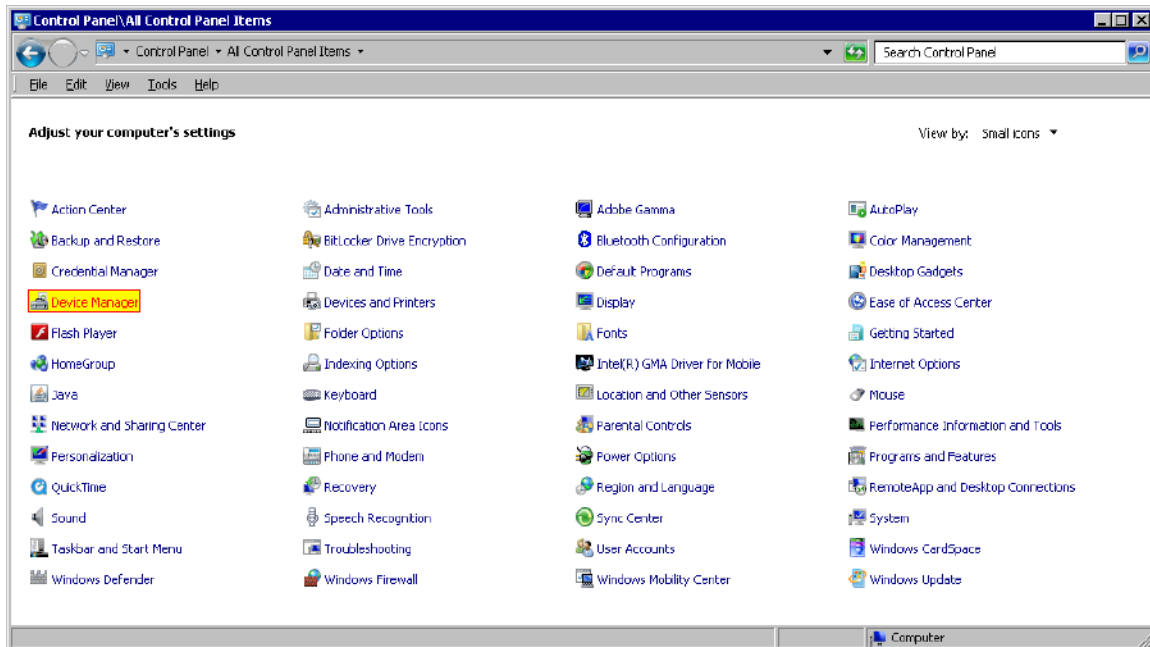
Windows 7

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

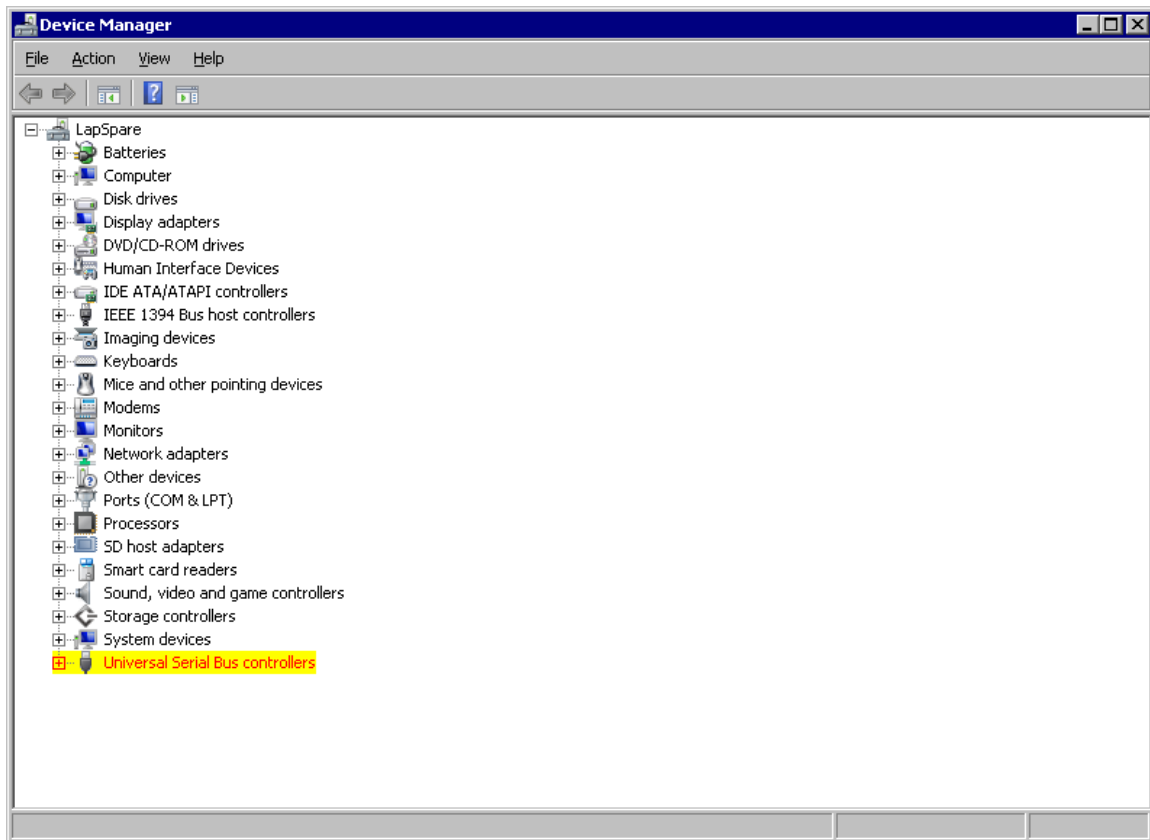
1. Click **Start ► Control Panel**. Windows displays the Control Panel screen.



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

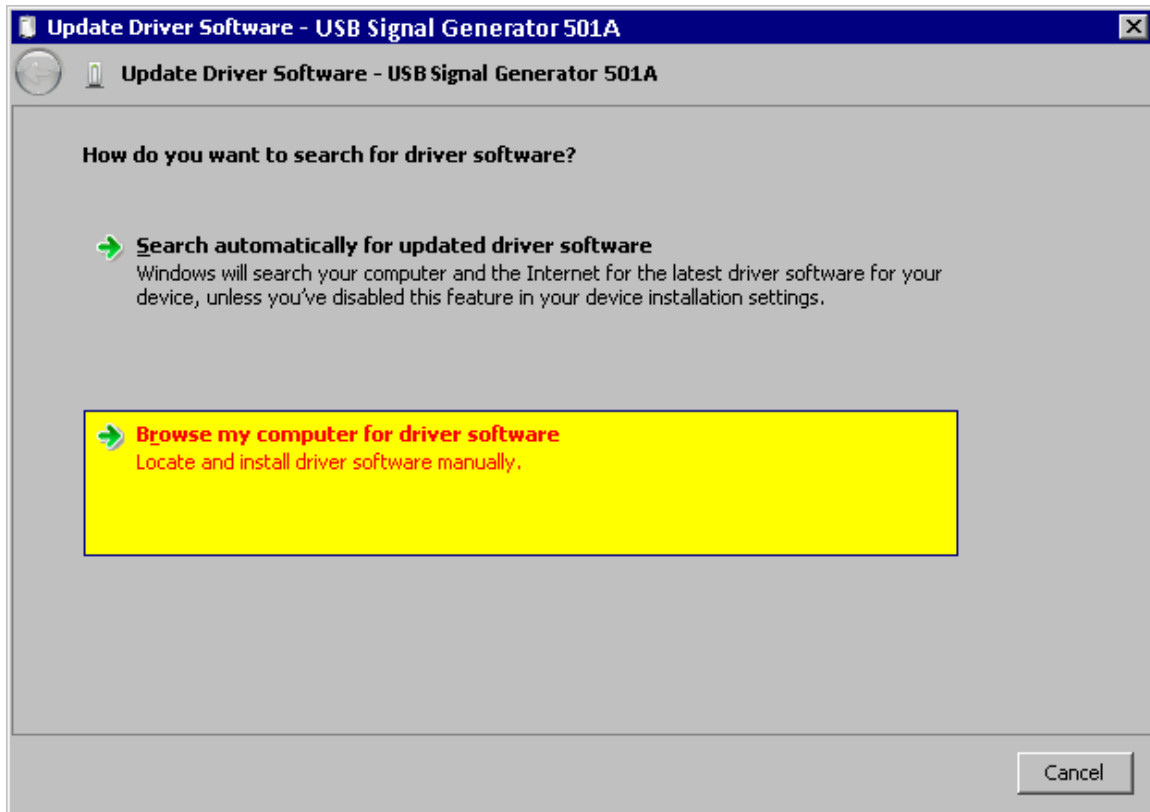


3. Click **Device Manager**. Windows displays the Device Manager screen.



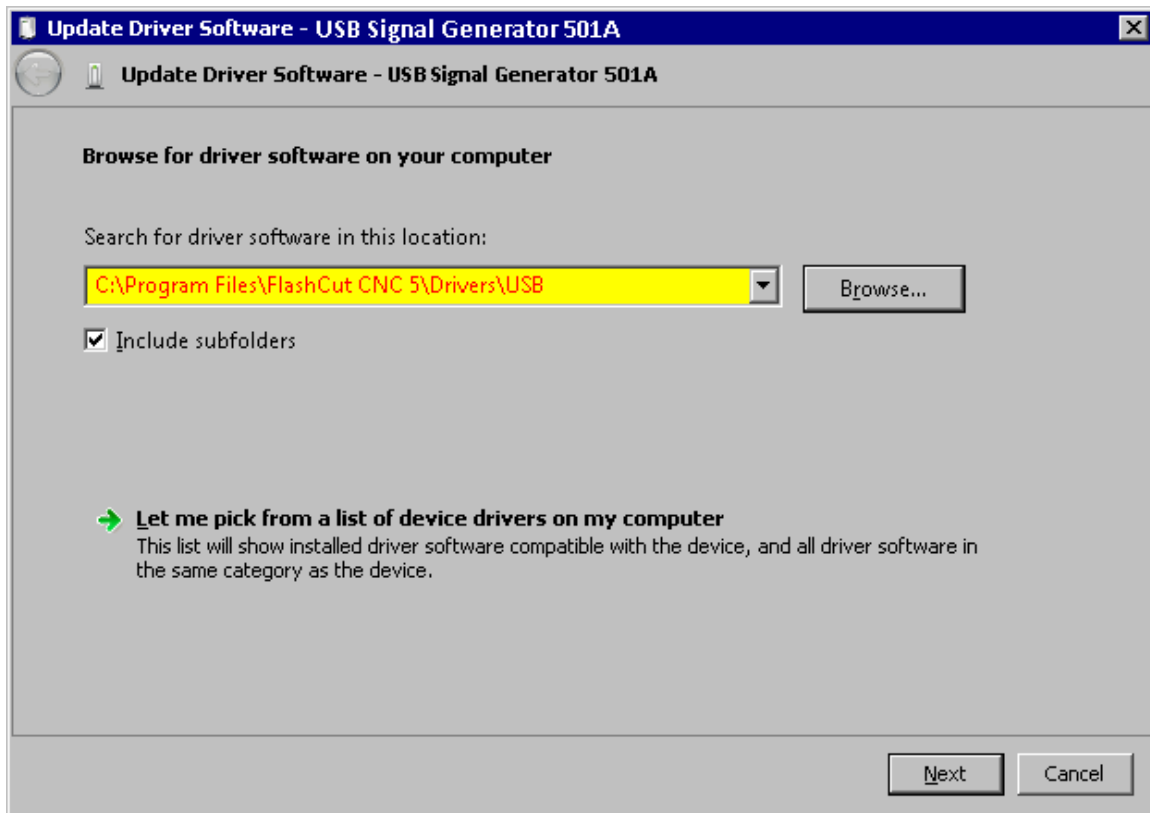
4. Expand the **Universal Serial Bus Controllers** listing.

5. Right click on **USB Signal Generator 501A** and select **Update Driver Software**. Windows displays this dialog box.



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

6. Windows displays the following dialog box.

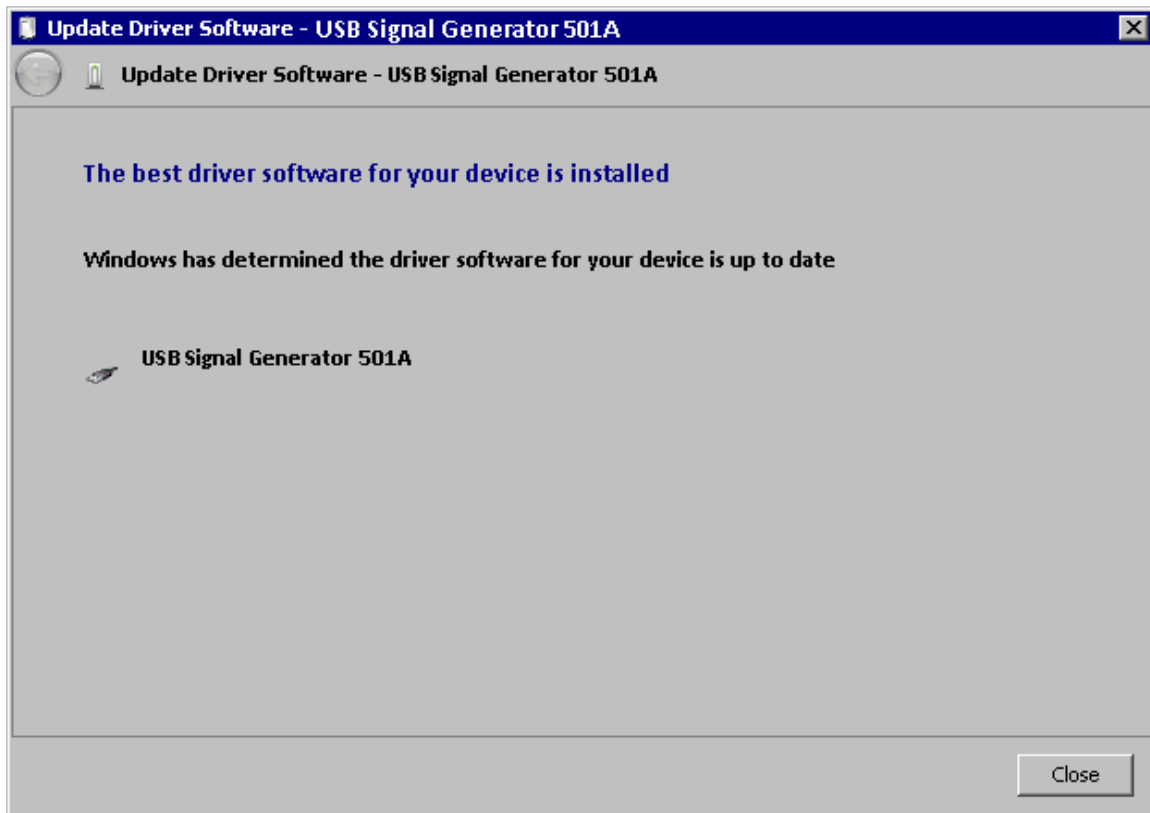


7. Click **Browse** and navigate to the following folder:

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

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

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

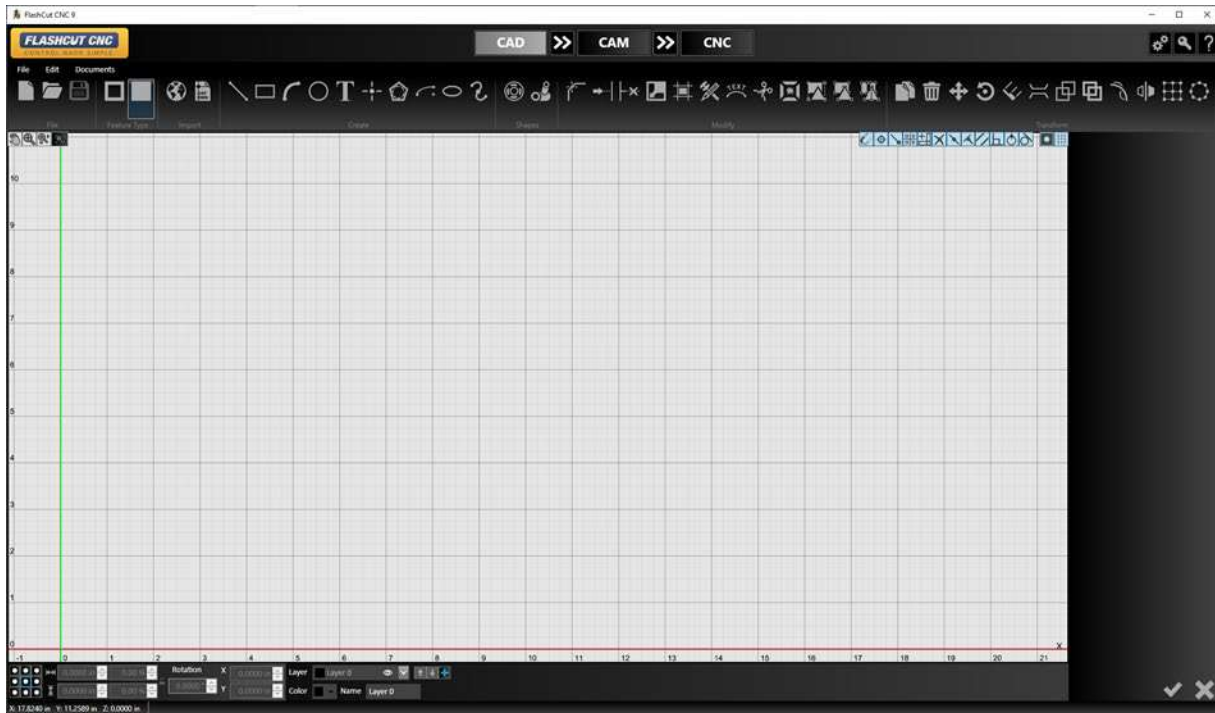


Click **Close**.

Getting started with FlashCut CNC

Launching the software

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



Running in evaluation mode

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

Activating a license



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

Configuring the software



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

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

Getting help



To view the program version, firmware, graphics driver status, select the **Help** button in any workspace. From the Help window, you may also view this user's guide, or the Programming Reference.

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

Understanding common interface elements

Configuration, License, and Help buttons



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

- The **Configuration** interface modifies program settings stored in the setup file. for more information.
- The **License** interface is used to install license files, and activate or deactivate program licenses. See [Licensing FlashCut](#) for more information.
- The **Help** interface is used display version numbers, and to build a support file for use during troubleshooting. See [Getting Help](#) for more information.



Tabs

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

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

FlashCut.CAD

arrows send to CAM and create toolpath

FlashCut.CAM

arrows send toolpath to CNC and generate G-Code

FlashCut.CNC



Accept / Cancel

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

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



Menu bar

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

- **File menu**
- **Edit menu**
- **Documents menu**



File menu

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

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



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

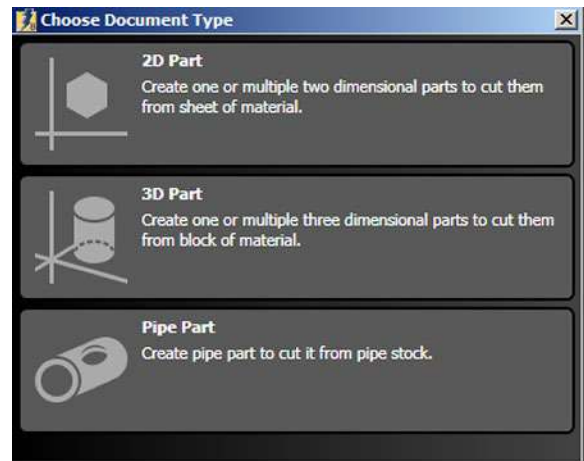
- **Print.Labels**
- **Print.Nest**

New (Ctrl+N)

Creates a new CAD/CAM drawing in a new FlashCut CAD window.

When selected from the menu or entered from the keyboard, the **New** file command displays a dialog window that requires the user to select the type of file: **2D Part**, **3D Part**, or **Pipe Part**.

The command does not close the current drawing. Note that you can view any open drawing by selecting it from the **Documents menu**.



Open (Ctrl+O)

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

Close

Closes the current CAD/CAM drawing from the CAD editor. If the drawing is unsaved, the software prompts you to save the file before closing.

Save (Ctrl+S)

Saves the current CAD/CAM drawing using the existing file name and location. It will not save any other CAD/CAM drawing that is open. FlashCut will prompt you to create a file name and destination. FlashCut saves drawings to the CAD/CAM (file extension .cadcam) format. These files are readable by FlashCut version 7 or later. The Save command is unavailable when no recent changes have been made.

Save as

Saves the current drawing to a new file name or destination. It will not save any other CAD/CAM drawing that is open. FlashCut prompts you to create a file name and destination. Saved files use the extension .cadcam. These files are readable by FlashCut version 7 or later.

Save all

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

Export

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

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

- **Export DXF**

Brings up the Windows save dialog. No parameters can be altered. The available **file formats** are DXF 2000 or DXF R12.

- **Export DXF with Options**

Brings up the Windows dialog box, and provides a configuration panel in the **Parameters area**. After setting the parameters for the file, you may either **Accept** or **Cancel** the changes.



- **Export Curves as Polylines**

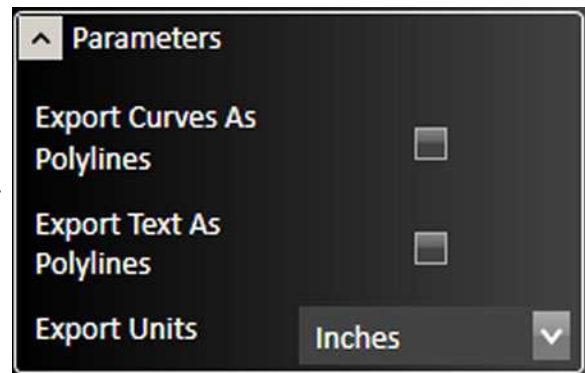
When selected, curves will be saved as polylines in the DXF file. When not selected, arcs and circles will be maintained, but ellipses and splines will be saved as polylines. Polylines are drawing objects composed of multiple separate line segments.

- **Export Text as Polylines**

When selected, text shapes will be saved as polylines in the DXF file. Polylines are drawing objects composed of multiple separate line segments. When not selected, text will be saved as a font.

- **Export Units**

Choose from either inches or millimeters



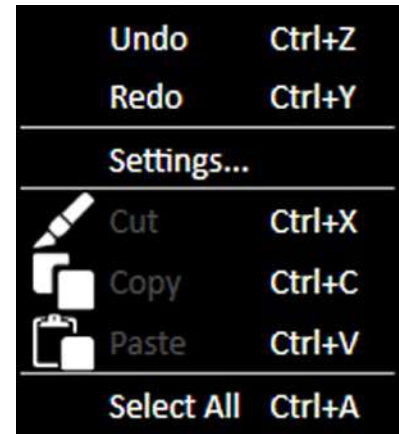
Exit

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

Edit menu

The **Edit** menu has the following commands:

- **Undo** (Ctrl+Z)
- **Redo** (Ctrl+Y)
- **Settings...**
- **Cut** (Ctrl+X)
- **Copy** (Ctrl+C)
- **Paste** (Ctrl+V)
- **Select All** (Ctrl+A)



Undo (Ctrl+Z)

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

Redo (Ctrl+Y)

Repeats the previous drawing action, or reverses the Undo stack.

Settings...

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



The screenshot shows the 'Settings...' dialog box with two sections: 'Grid' and 'Units'. In the 'Grid' section, 'Visible' and 'Dynamic Grid' are both checked. 'Grid Major Spacing' is set to '1.0000 in' and 'Sections Per Cell' is set to '4'. In the 'Units' section, 'Document Units' is set to 'in'.

Grid

- **Visible**

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

- **Dynamic Grid**

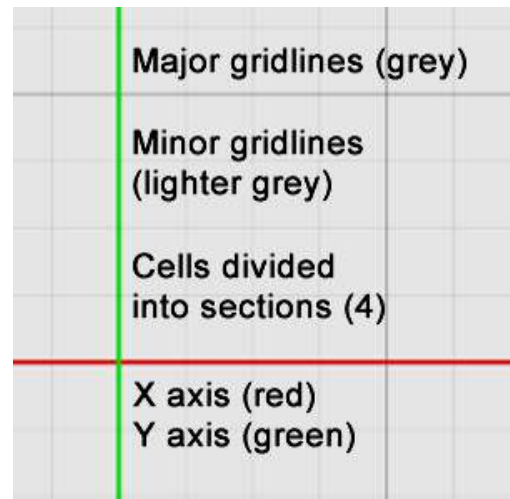
Toggles dynamic gridlines within the drawing window. When checked, these gridlines remain the same size on the screen despite zooming and panning motions. When unchecked, the dimension that these gridlines represent remain the same while zooming and panning.

- **Grid Major Spacing**

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

- **Sections Per Cell**

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



Units

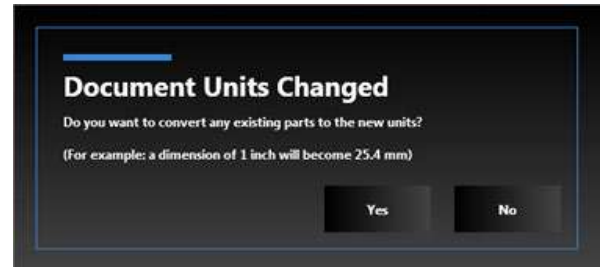
- **Document Units**

Select the dimensional units of the CAD drawing. You may choose the following units:

- mm – millimeters
- in – inches

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

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



Cut (Ctrl+X)

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

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



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

Copy (Ctrl+C)

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



Paste (Ctrl+V)

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



Select All (Ctrl+A)

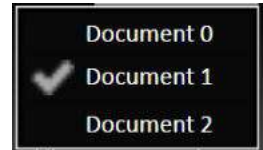
Use the **Select All** command to select all geometry in the workspace.

Documents menu

The **Documents** menu allows you to toggle between all open drawings in both **FlashCut CAD** and **FlashCut CAM**.

The checked document is visible and available for editing.

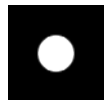
Note You may copy features from one drawing and paste them into another drawing.



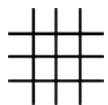
Part shading and Grid tools

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

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



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



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

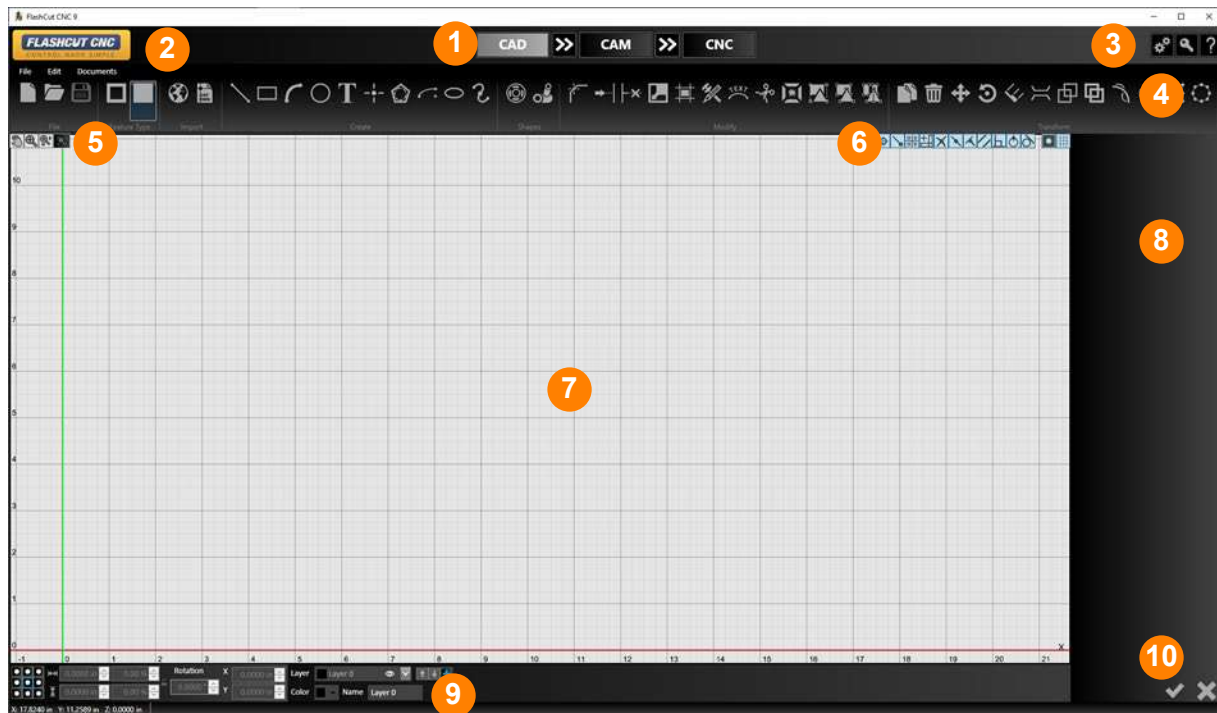


Reports the real time position of the cursor. In the **CAD** window, provides instructions to the user for the expected input of the currently selected tool.

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.

The FlashCut CAD main screen is shown here.



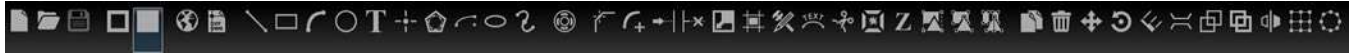
An explanation of each area of the screen is provided in these topics.

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

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

Ribbon

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



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

- **File tools**
- **Feature.Type**
- **Create tools**
- **Shapes**
- **Transform tools**

File tools

There are three file tools on the ribbon:

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

New (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 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 the 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. 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.

This option is selected by default.

Import tools

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

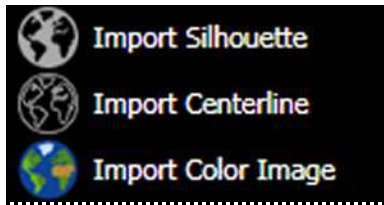
- *.bmp
- *.dib
- *.jpg
- *.jpeg
- *.jpe
- *.png
- *.pbm
- *.pgm
- *.ppm
- *.sr
- *.ras
- *.tiff
- *.tif

There are four ways to import a drawing:

Import silhouette, centerline, or color image



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



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

Use Arc Fitting

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

- **Fitting Tolerance**

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

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

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

Import DXF/DWG file



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

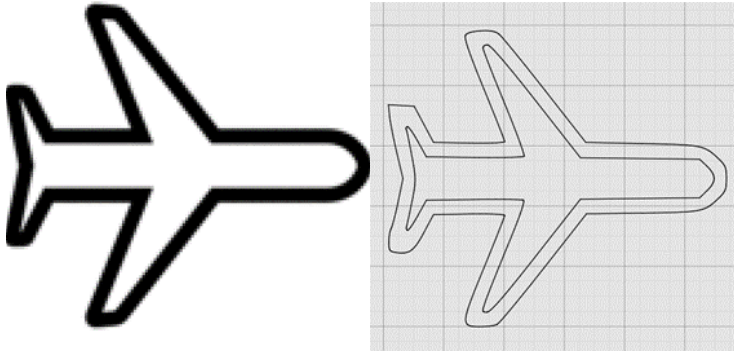
Understanding silhouette and centerline images

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 35.

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.

General

Choose Image

[choose file] **Browse...**

Scale

1.0000 mm 100.00 %

1.0000 mm 100.00 %

Position X

0.0000 mm

Position Y

0.0000 mm

Select All

☐

Create Group

☒

Display Preview Image

☒

Progress

Create Silhouette From Photo

Enable

☐

Tracing Basic Parameters

Inverted

☐

Intensity Threshold

0% 100%

40.00 %

Segmentation Quality

1% 100%

20.00 %

Tracing Advanced Parameters

Detail Level

0% 100%

100.00 %

Despeckling (sq. pix)

3

Arc Fitting

Use Arc Fitting

☐

Fitting Tolerance

0.0010 mm

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

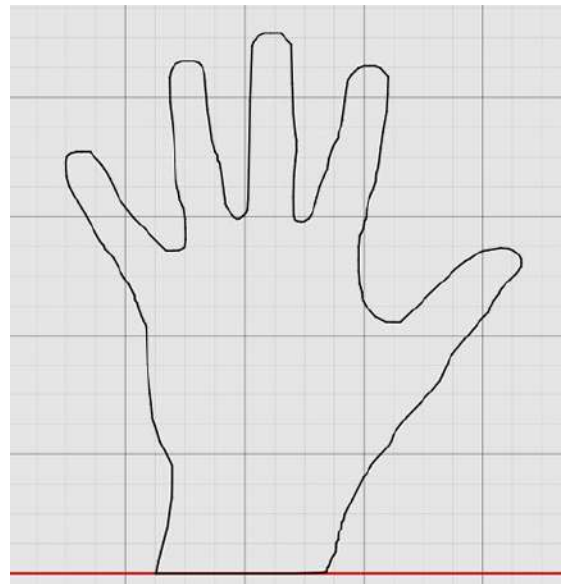
Superimposes a translucent preview of the imported image while the new drawing is being generated.

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:

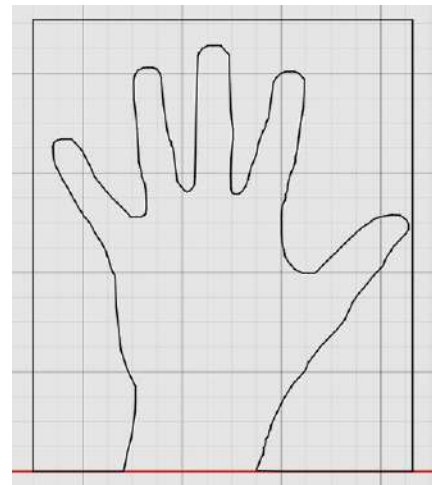
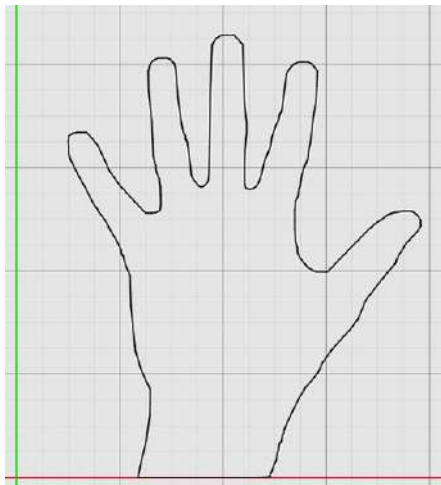
- **Background color** –Select which corner of the image that is to be used to sample the background color. The color sampled from the specified corner will be set as the background color. When importing a photo, the edges of the object are determined by the color contrast between an object and the selected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

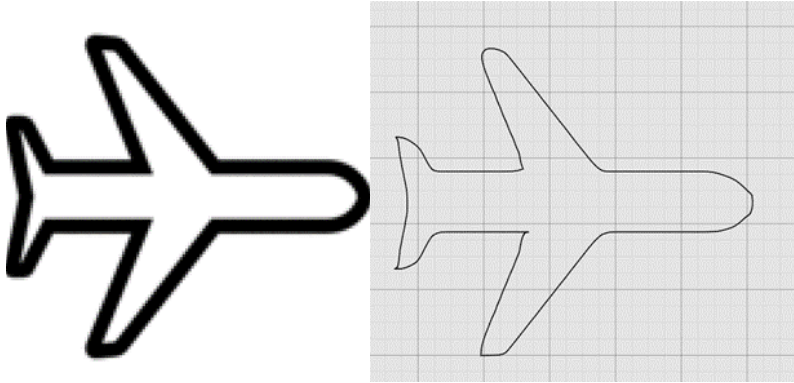
When selected, **Arc Fitting Tool** will be applied to the imported image.

- **Fitting Tolerance**

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

Import centerline image

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



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



Choose image

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

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.

Parameters

Choose Image

Scale

Position X

Position Y

Segmentation Quality

Maximum possible line width (pixels)

Inverted

☐

Select All

☐

Create Group

☒

Display Preview Image

☒

Progress

Arc Fitting

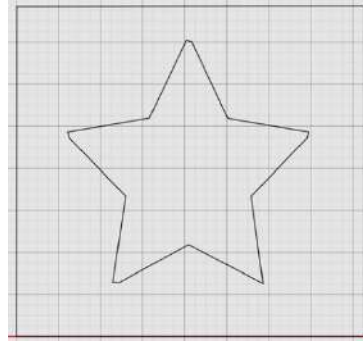
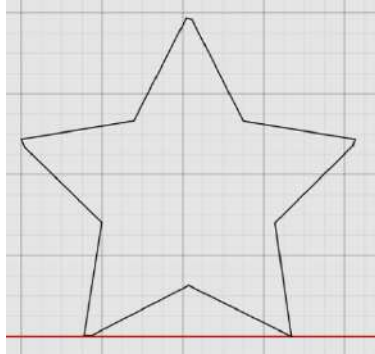
Use Arc Fitting

☐

Fitting Tolerance

Inverted

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



Select all

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

Create Group

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

Display Preview Image

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

Progress

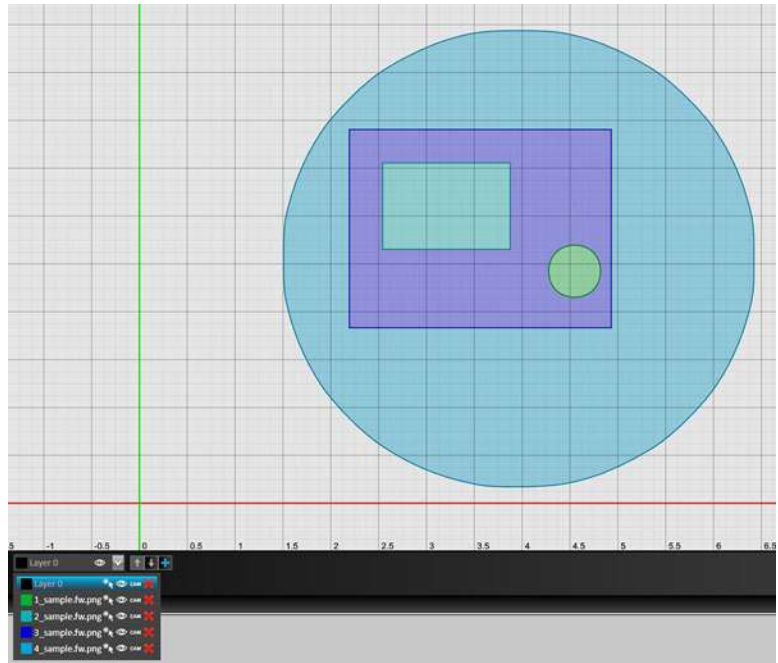
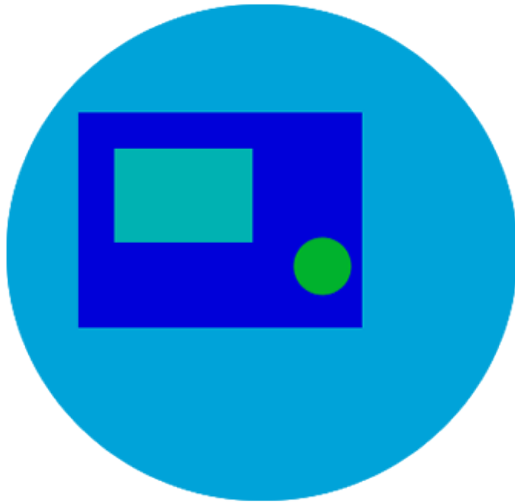
Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for rendering to finish between edits.

Arc Fitting

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

Import color image

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



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



Choose image

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

Scale

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

Position X

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

Position Y

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

Select all

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

Create Group

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

Display Preview Image

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

Progress

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

Color Tolerance

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

Despeckling

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

Segmentation quality

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

Arc Fitting

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

The screenshot displays the settings panel for FlashCut CAD/CAM, organized into four sections:

- General**:
 - Choose Image**: Includes a "[choose file]" button and a "Browse..." button.
 - Scale**: Features two rows of input fields. The first row shows "0.9069 in" and "35.29 %". The second row shows "1.0000 in" and "35.29 %".
 - Position X**: Input field set to "1.0000 in".
 - Position Y**: Input field set to "0.0000 in".
 - Select All**: A checkbox that is currently unchecked.
 - Create Group**: A checkbox that is checked.
 - Display Preview Image**: A checkbox that is checked.
 - Progress**: A horizontal progress bar.
- Trace Color Image**:
 - Color Tolerance**: A slider ranging from 0% to 100%, with a value of 3.00 % displayed.
 - Despeckling (sq. pix)**: An input field set to 3.
 - Segmentation Quality**: A slider ranging from 1% to 100%, with a value of 20.00 % displayed.
- Arc Fitting**:
 - Use Arc Fitting**: A checkbox that is currently unchecked.
 - Fitting Tolerance**: An input field set to 0.0010 in.

Import DXF/DWG image

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



Choose image

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

Scale

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

Imported layers

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

Position X

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

Position Y

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

Select All

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

Create Group

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

Combine Layers

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

Progress

Shows progress of file import.

The 'Parameters' dialog box contains the following elements:

- Choose DXF/DWG file:** A section with a '[choose file]' button and a 'Browse...' button.
- Scale:** A numeric input field set to '100.0000 %' with up/down arrow buttons.
- Imported Layers:** A list box showing imported layers.
- Position X:** A numeric input field set to '0.0000 in' with up/down arrow buttons.
- Position Y:** A numeric input field set to '0.0000 in' with up/down arrow buttons.
- Select All:** A checkbox that is currently unchecked.
- Create Group:** A checkbox that is currently checked.
- Combine Layers:** A checkbox that is currently unchecked.
- Progress:** A progress bar at the bottom of the dialog.

Create tools

These tools add new elements to the drawing.

After selecting a tool, move the cursor into the drawing window in order to begin constructing the element.



You may use the mouse to place and size each element or you can edit parameters for the element, such as dimensions and location, in the parameter window after creating the element. Click the shape or features and then edit the desired parameters.

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



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



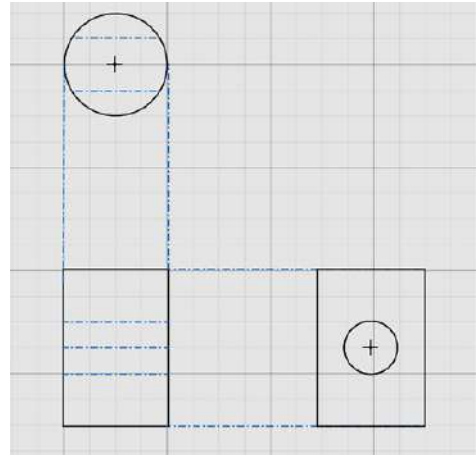
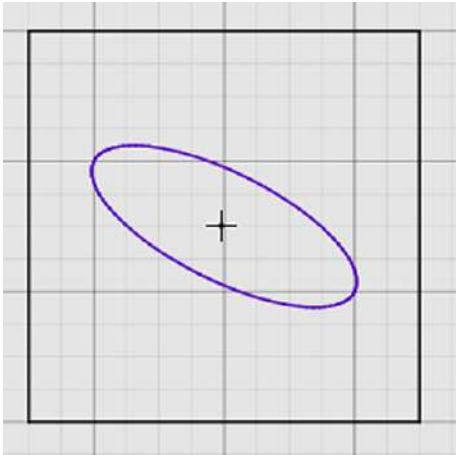
Select **Cut Element** to use the default cutting Fab Head. This is the pre-selected option. These elements appear in the CAD window as solid black lines.



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

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

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



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

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

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

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

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**.

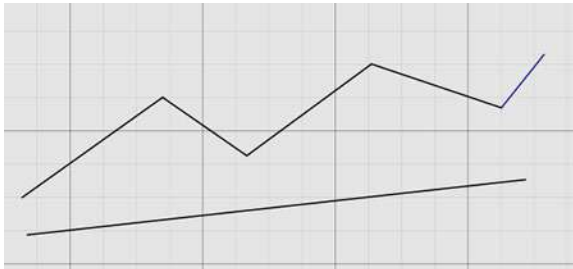
When creating a Two Point Line, the properties may be adjusted manually in the parameters window. Any single line segment (e.g., any segment that is part of a polygon or a multi-segment line) may be selected and its properties displayed.

Creation mode

Select the type of line to create.

- **Continuous Line**
- **Two Point Line**

Examples of each type are shown below.



Position, offset, angle and length

Specify the attributes of a specific line segment.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the line a construction line.

^ Parameters

Creation mode

 Continuous Line

 Two Point Line

Start Position X 0.0000 in

Start Position Y 0.0000 in

End Position X 0.0000 in

End Position Y 0.0000 in

X Offset 0.0000 in

Y Offset 0.0000 in

Angle 0.0000 °

Length 0.0000 in

Treatment

 Cut Element

 Mark Element

For Construction ☐

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.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the rectangle a construction line.

The screenshot shows the 'Parameters' panel for the Rectangle tool. It includes a 'Creation mode' section with five options: 'Corner Rectangle' (selected), 'Center Rectangle', '3 Point Corner Rectangle', '3 Point Center Rectangle', and 'Parallelogram'. Below this are input fields for 'Position X', 'Position Y', 'X Offset', and 'Y Offset', each with a value of '0.0000 in'. The 'Treatment' section has two options: 'Cut Element' (selected) and 'Mark Element'. At the bottom, there is a 'For Construction' checkbox.

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.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the arc a construction line.

Parameters

Creation mode

Centerpoint Arc

Tangent Arc

3 Point Arc

Center Position X

0.0000 in

Center Position Y

0.0000 in

Radius

0.0000 in

Start Position X

0.0000 in

Start Position Y

0.0000 in

Start Angle

0.0000 °

End Position X

0.0000 in

End Position Y

0.0000 in

End Angle

0.0000 °

Treatment

Cut Element

Mark Element

For Construction

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

Examples of each type are shown below.

Position, radius, and diameter

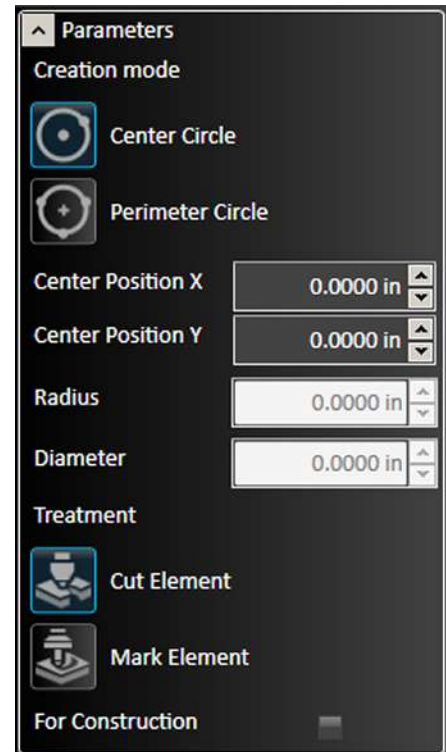
Specify the attributes of the circle.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the line a construction line.



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.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the text object a construction line.

The Parameters panel for text objects includes the following settings:

- Position X:** 4.5000 in
- Position Y:** -2.0000 in
- Text:** Text
- Font Name:** Arial
- Alignment:** Left
- Size:** 1.0000 in
- Kerning Offset:** 0.0000
- Line Spacing:** 135.0000 %
- Angle:** 0.0000 °
- Bold:** ☐
- Italic:** ☐
- Treatment:**
 - ☒ Cut Element
 - ☐ Mark Element
- For Construction:** ☐

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.

Parameters	
Position X	11.0000 in
Position Y	-6.0000 in

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. Minimum number of sides is three.

Internal Diameter

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

External Diameter

Diameter of an exterior circle tangent to all vertices.

Side Length

Length of each side.

Parameters	
Center Position X	8.4073 in
Center Position Y	7.1130 in
Sides Count	6
Internal Diameter	2.4180 in
External Diameter	2.7921 in
Side Length	1.3961 in
Leave Construction Circle	☒
Treatment	
	Cut Element
	Mark Element
For Construction	☐

Leave Construction Circle

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

Treatment

Select whether the element should be cut or marked.

For Construction option

Select to make the polygon a construction line.

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.

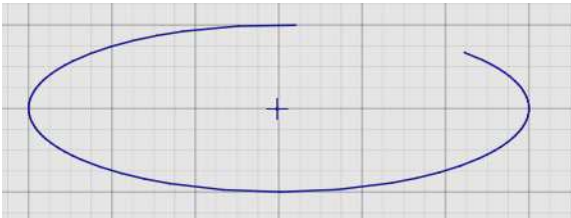
Finally, select a start point and then an end point to the arc.

Center Position X and Y

Specify the coordinates of each center point

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.



Major Axis Angle

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

Start and End Angle

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

Treatment

Select whether the element should be cut or marked.

For Construction option

Select to make the line a construction line.

Parameters	
Center Position X	12.6000 in
Center Position Y	13.2000 in
Radius 1	0.2000 in
Major Axis Angle	90.0000 °
Radius 2	0.6000 in
Start Angle	16.7307 °
End Angle	78.6901 °
Treatment	
	Cut Element
	Mark Element
For Construction ☐	

Ellipse



Creates an ellipse.

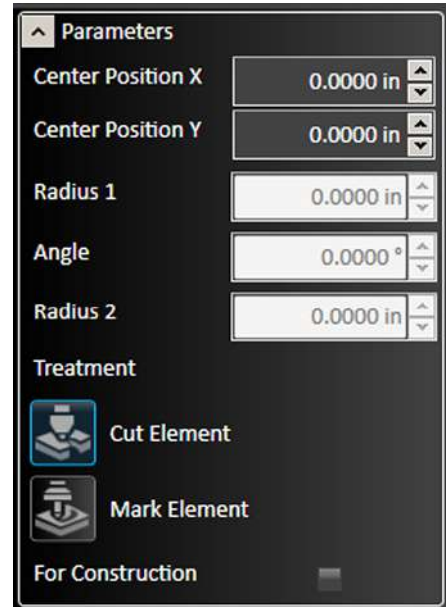
Drawing the ellipse

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.



The Parameters dialog box for the Ellipse tool contains the following settings:

Parameters	
Center Position X	0.0000 in
Center Position Y	0.0000 in
Radius 1	0.0000 in
Angle	0.0000 °
Radius 2	0.0000 in
Treatment	
☒ Cut Element	
☐ Mark Element	
For Construction	☐

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.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the ellipse a construction line.

Spline

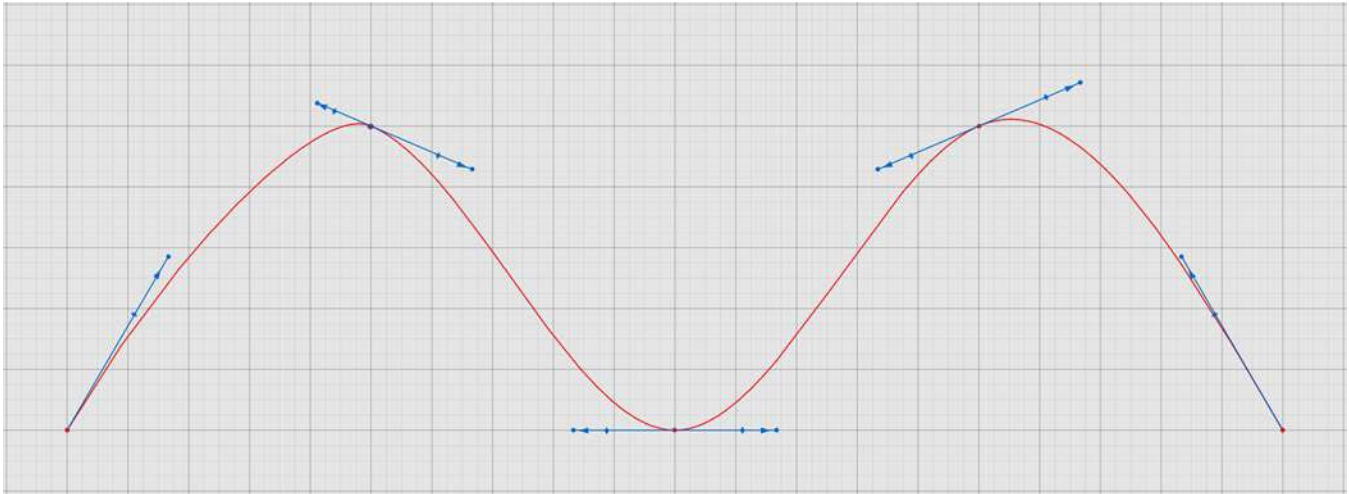


Creates a complex curve in multiple segments.

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.

Treatment

Select whether the element should be cut or marked.

For Construction

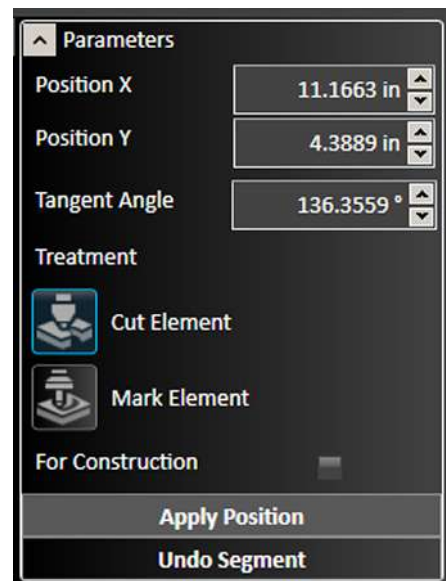
Select to make the line a construction line.

Apply Position

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

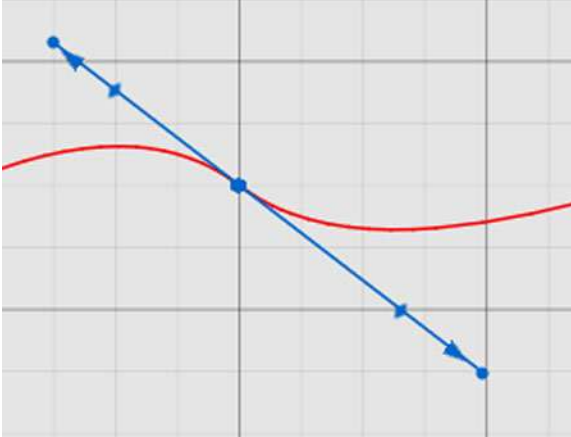
Undo Segment

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.

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the line a construction line.

Parameters	
Current Vertex	2
Position X	7.5000 in
Position Y	13.2500 in
First Magnitude	0.4747
Second Magnitude	0.6203
Angle	-37.4492 °
Treatment	
☒	Cut Element
☐	Mark Element
For Construction ☐	

Shapes

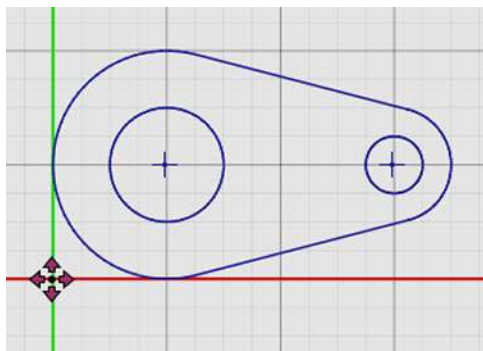
Shape tool



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



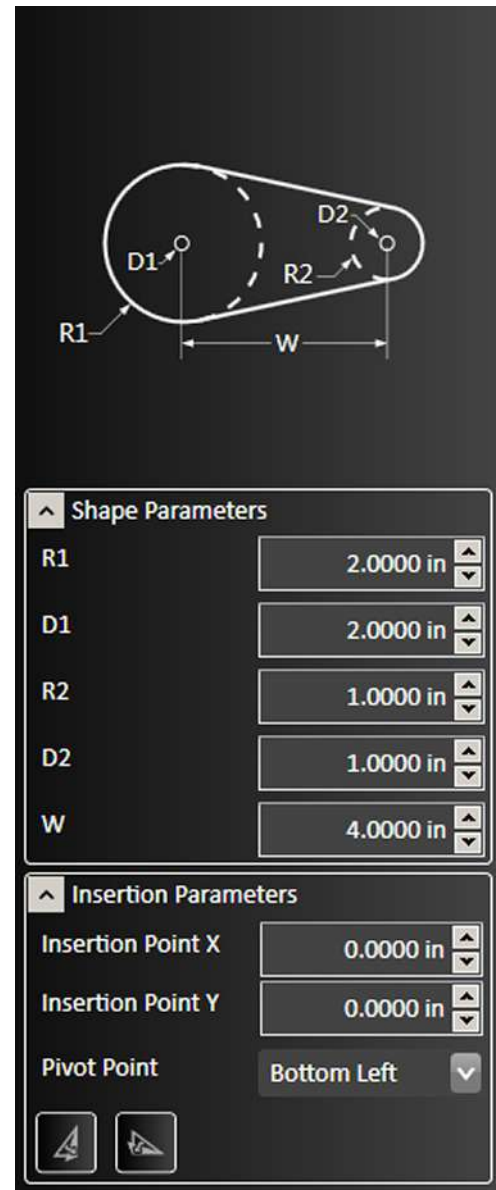
Shape Parameters

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.

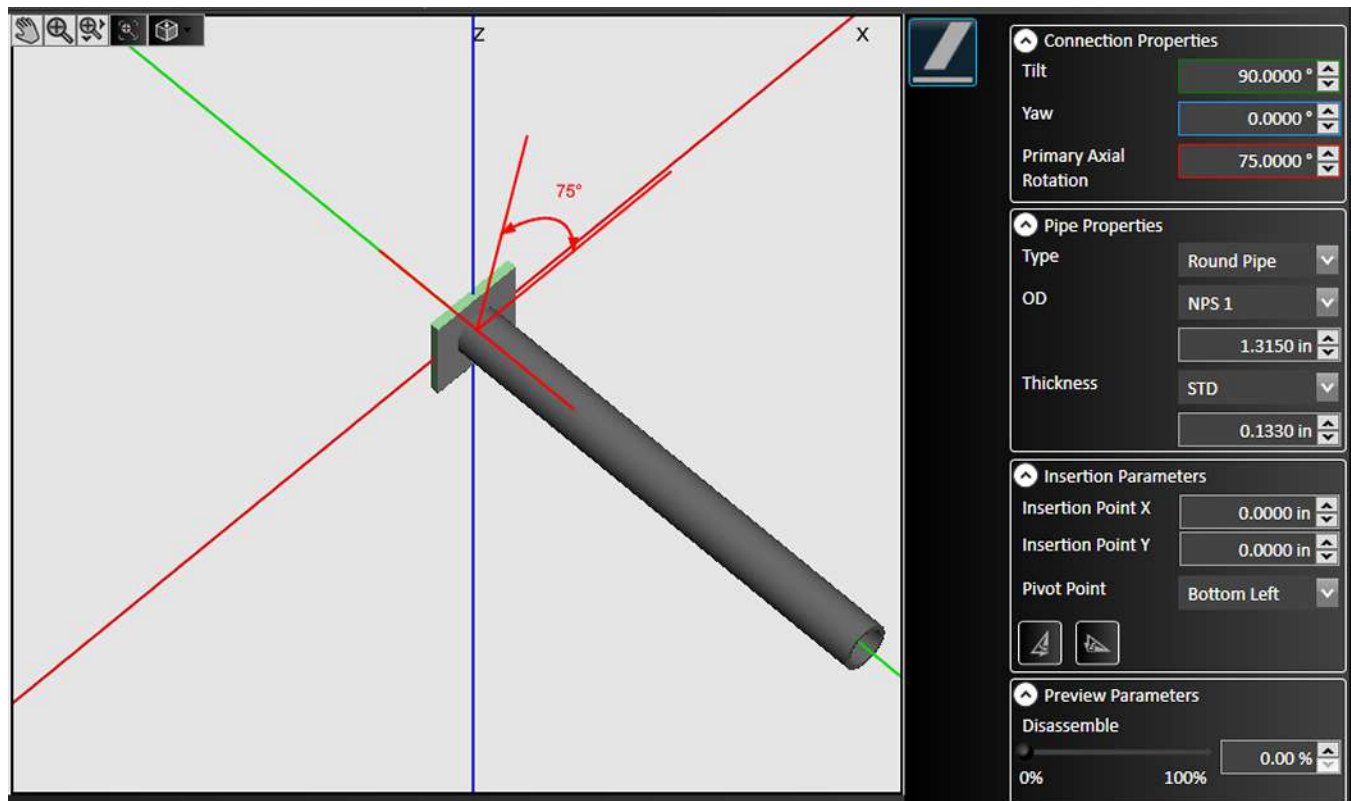
After changes are complete, you may either **Accept** or **Cancel** the changes.



Pipe Joint Shape Tool



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



Connection Properties

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

Pipe Properties

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

Insertion Parameters

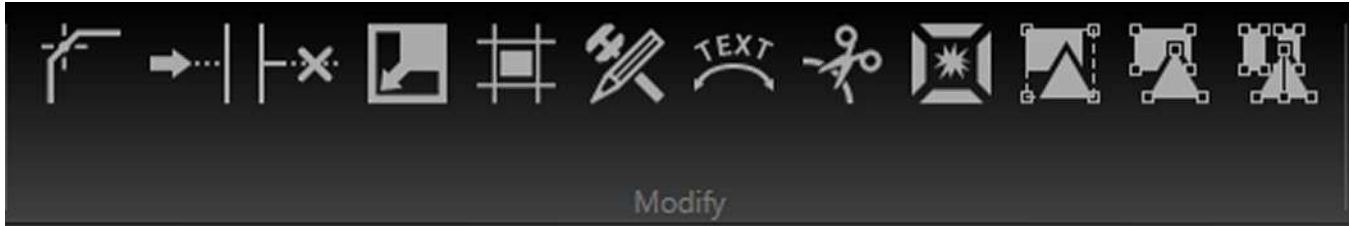
The position of the **X/Y Insertion Points** and the location of its **Pivot Point** can be specified. In addition, the shape can also be flipped horizontally or vertically.

Preview Parameters

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

Modify tools

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



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

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

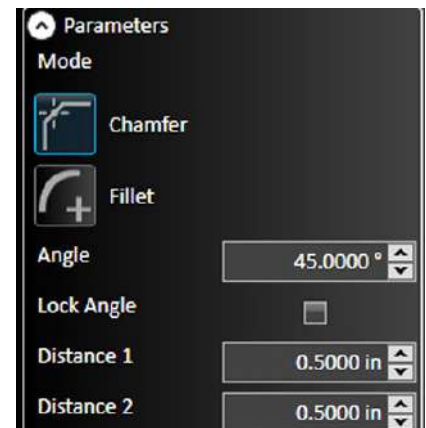
- **Corner.Tool**
- **Extend.Tool**
- **Trim.Tool**
- **Scale.Entities**
- **Apply.Offset**
- **Fix.Drawing**
- **Curve.Text**
- **Cut.Tool**
- **Explode.Entities**
- **Group**
- **Ungroup**
- **Ungroup.All**

Corner Tool



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

Apply Chamfer

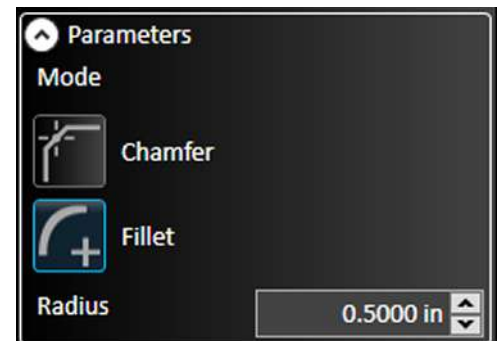


Select two intersecting lines or their intersection point, and then click **Chamfer** to add a sloped chamfer to this edge. You can scale the size of the chamfer by dragging the mouse towards or away from the chamfer, or in the parameter window, type the distances from the intersection in both directions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

Apply Fillet

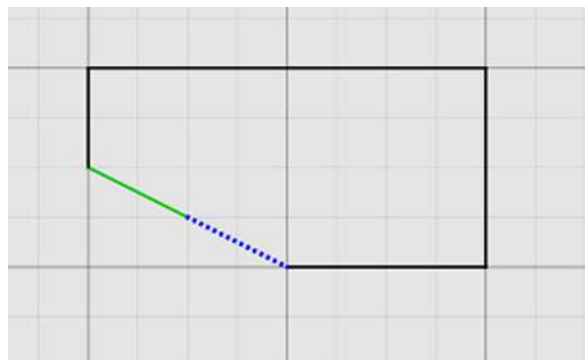
Select two intersecting lines or their intersection point, and then click **Fillet** to add a rounded fillet to this edge. You can scale the size of the fillet by dragging the mouse towards or away from the fillet, or in the parameter window, type in the radius of the fillet. Once the fillet is correctly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



Extend Tool



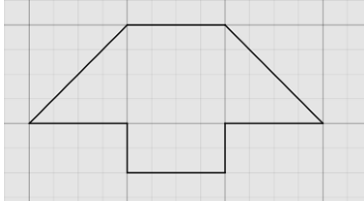
Click the **Extend Tool**, and then hover the selection cursor over the endpoint of the line or curve to be extended. A preview of the extension should appear (in blue, below) to show where the line or curve will intersect with another object in the drawing. Click the mouse to finalize the extension.



Trim Tool



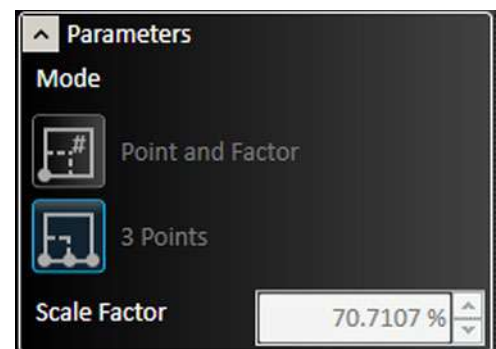
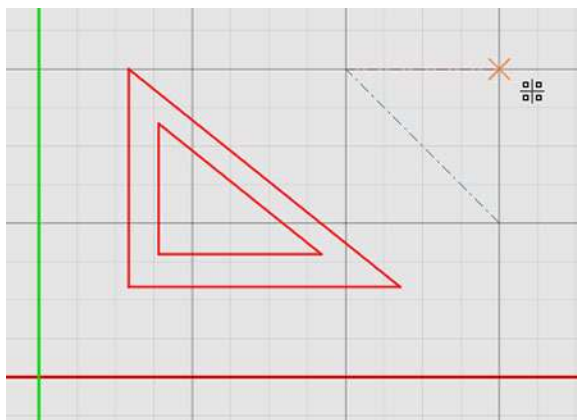
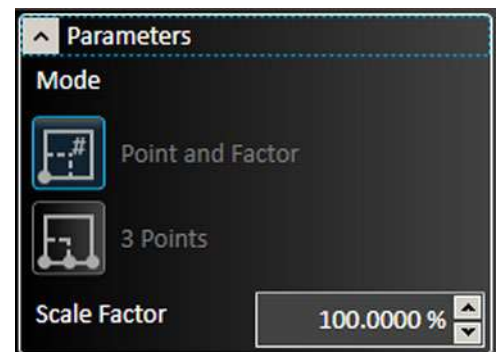
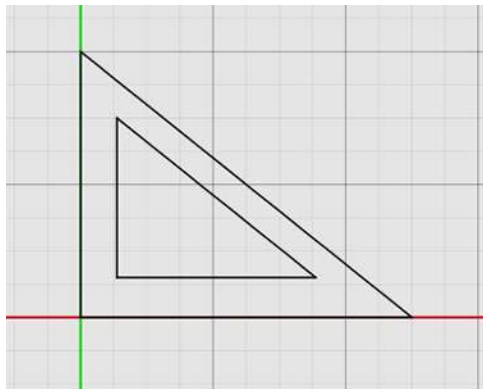
Click the **Trim Tool**, and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear (in red, below) where the line or curve will be removed. Click the mouse to finalize the trim.



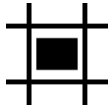
Scale Entities



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.

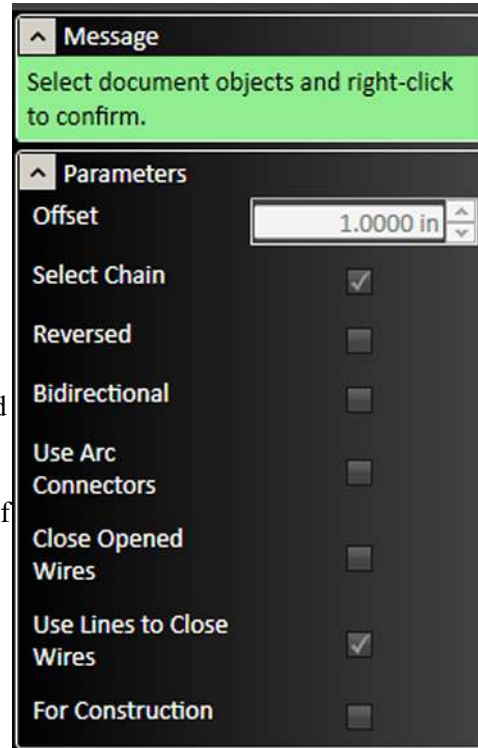
Offset

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.

Bidirectional

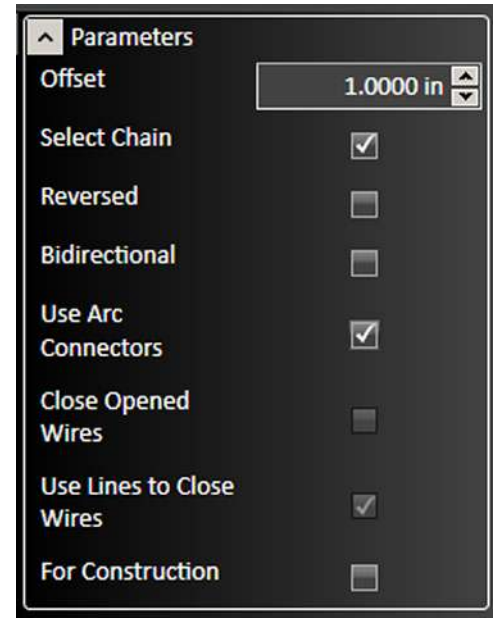
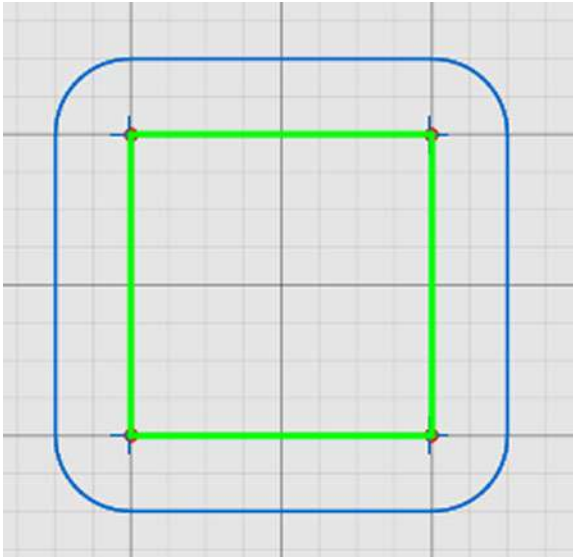
Bidirectional creates a second copy, opposite the first. When the **Bidirectional** option is chosen, the **Close Opened Wires** option appears.

Close Opened Wires

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

Use Arc Connectors

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



Use Lines to Close Wires

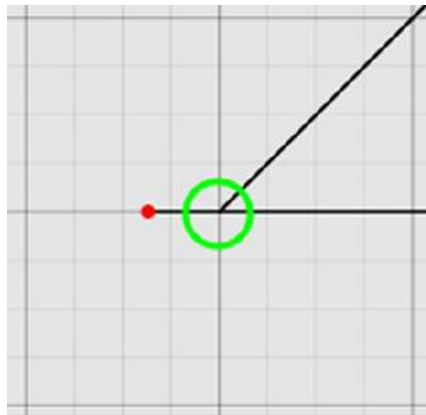
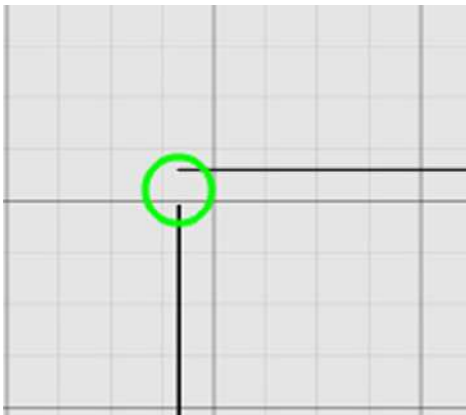
Use Lines to Close Wires changes the rounded edges to flat, straight lines.

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.

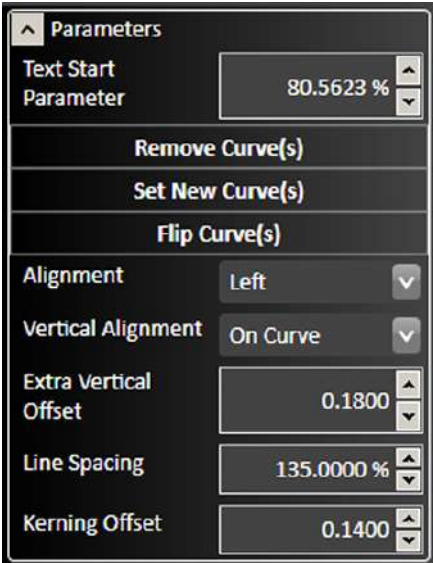
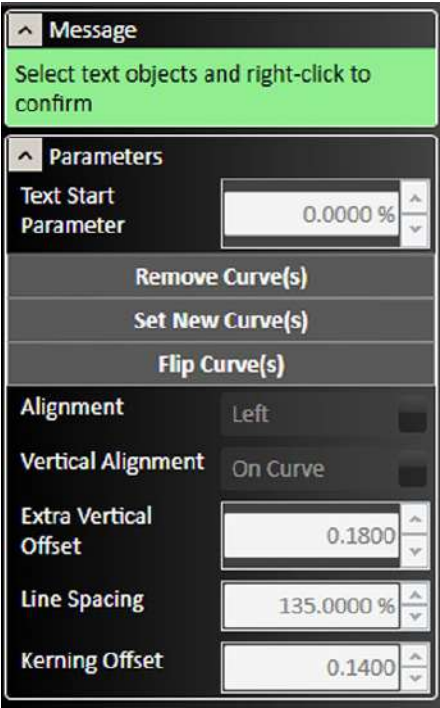
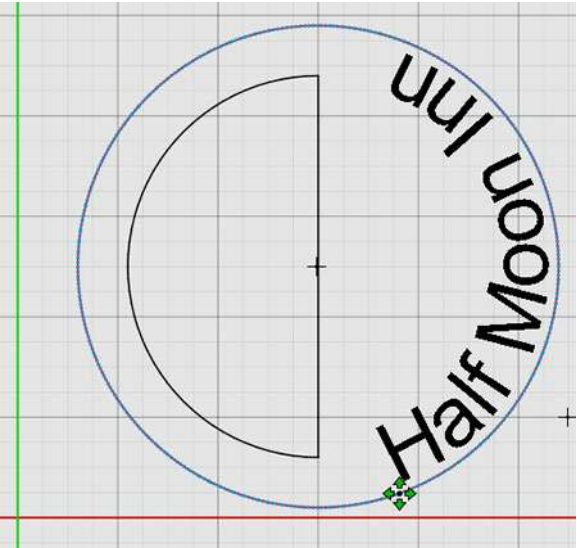
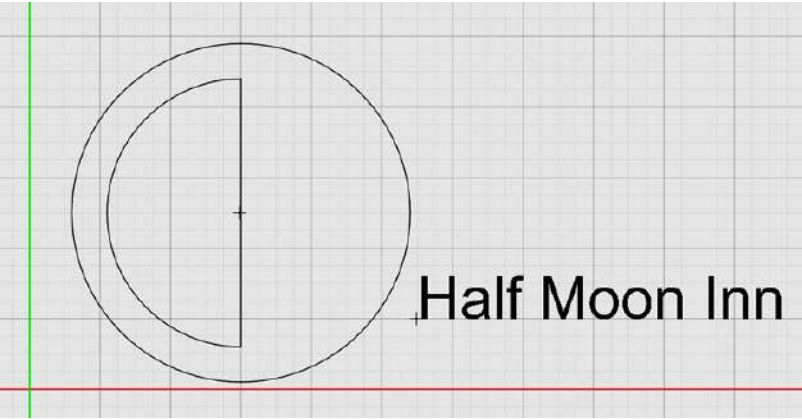
Curve Text



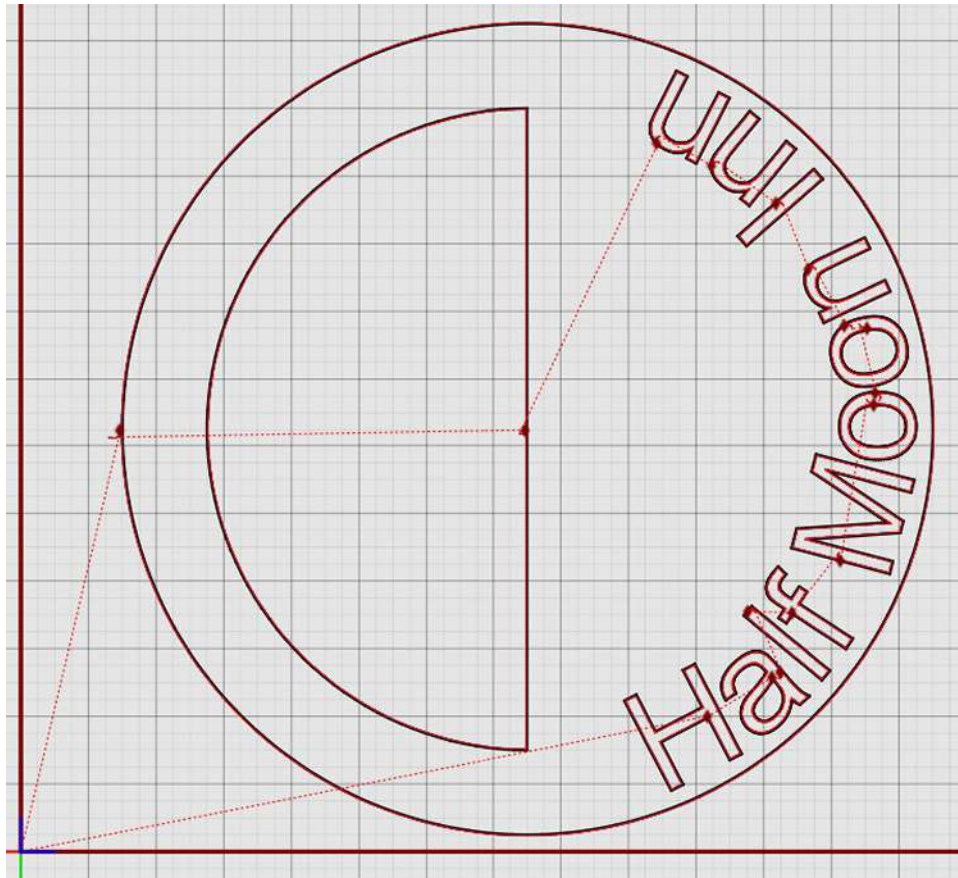
The **Curve Text** tool modifies an existing text object to follow a curved shape. Start by creating both the text object and the curved shape you want to use.

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.



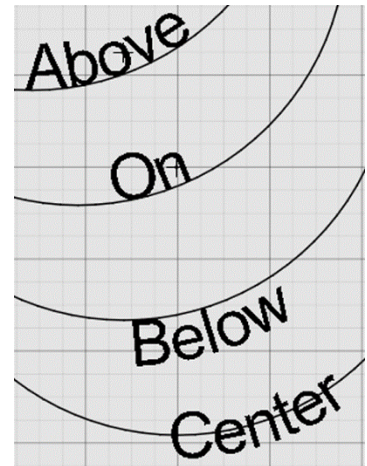
Alignment

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.

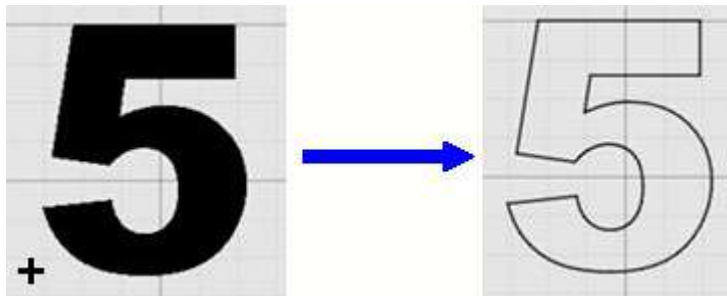
Select the **Cut Tool** and click the mouse at the position on the entity where you want the cut to be made. You can verify the cut by hovering the cursor over the feature. Only a part of it will highlight if the cut worked properly.

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

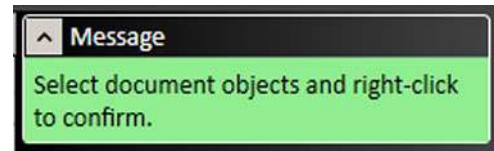
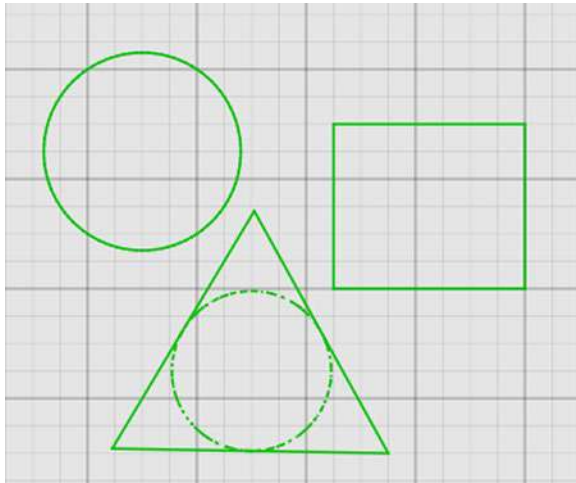
Ungroup All



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.

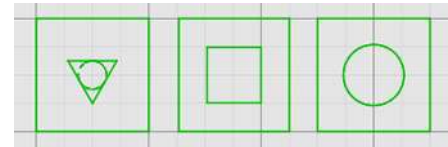
The example below shows the highlighting and message for all grouping operations.



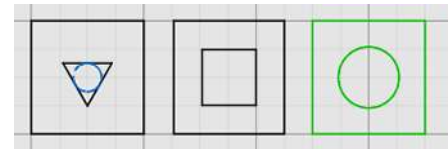
Ungroup vs. Ungroup All

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.

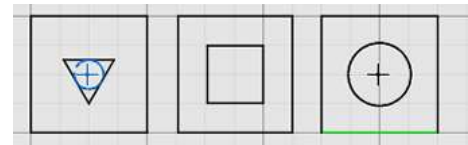
Each object in the group is itself a group of multiple objects.



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



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



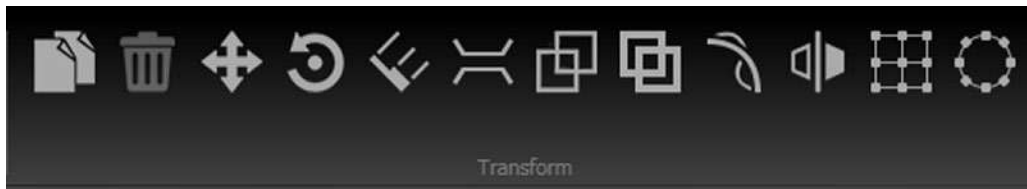
Transform tools

These tools do not change the physical properties of the drawing features. Instead, they modify the size, scale, and number of features in the drawing.

Select the features to be modified, and select the specific transform tool to use. You may edit parameters for the transformation in the parameter window. After transformations are complete, click the green check mark to

Accept changes or the red X mark to **Cancel** changes.  

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



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

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.

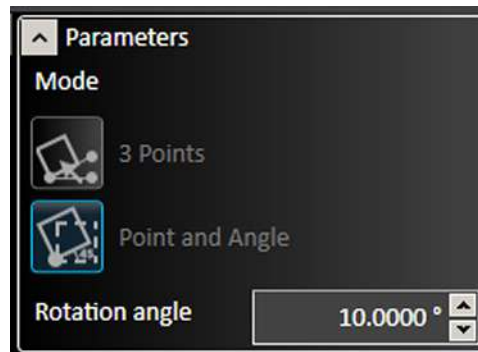
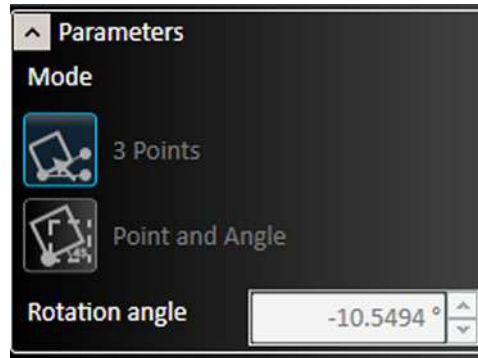
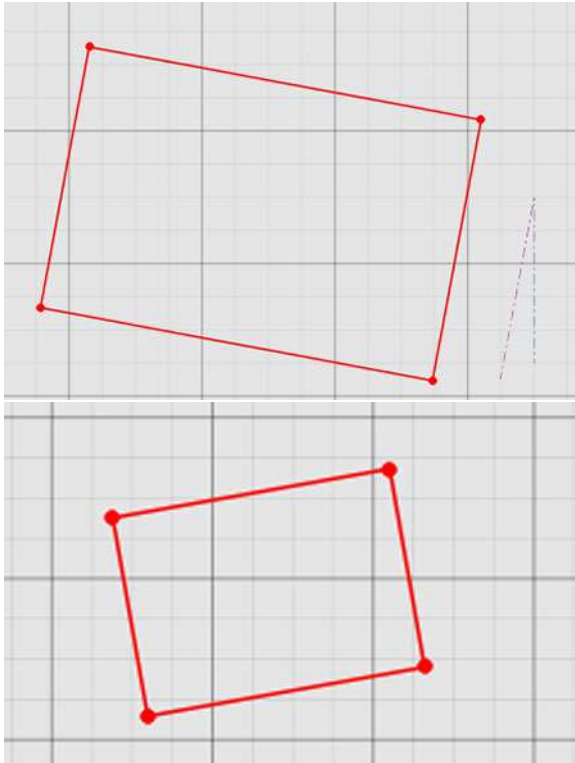
Rotate Entities



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.

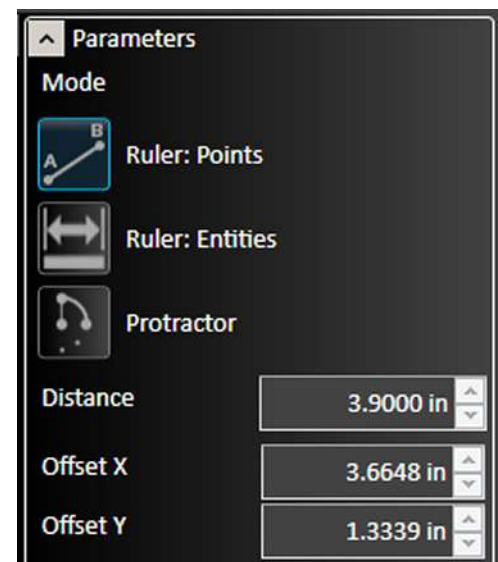
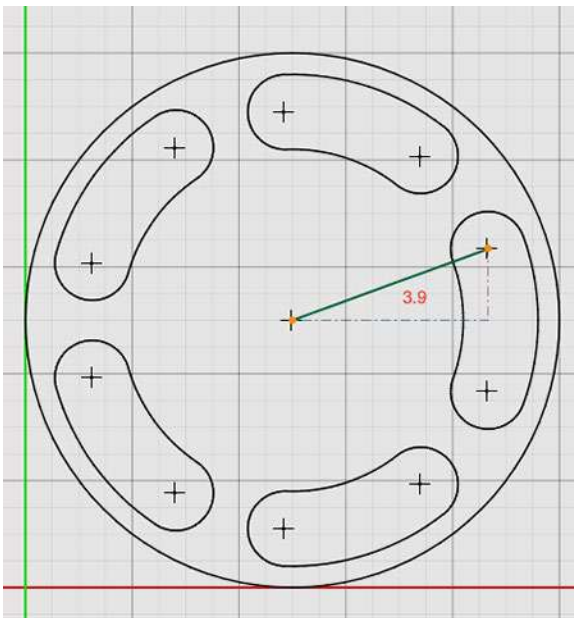
The **Point and Angle** rotation requires you to select a central rotation point, and then enter a rotation amount in degrees. After changes are complete, click the green check mark or press **Enter** to **Accept** changes or the red X mark to **Cancel** changes.



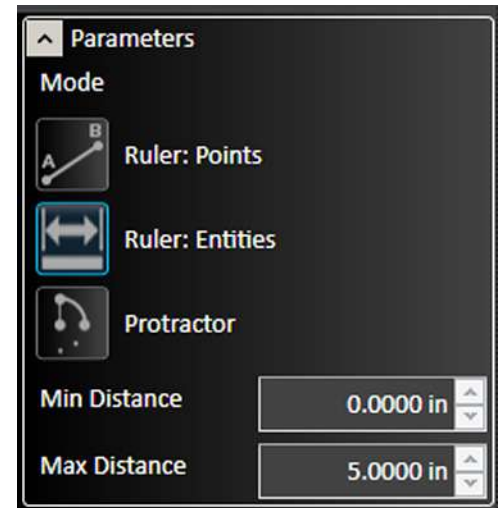
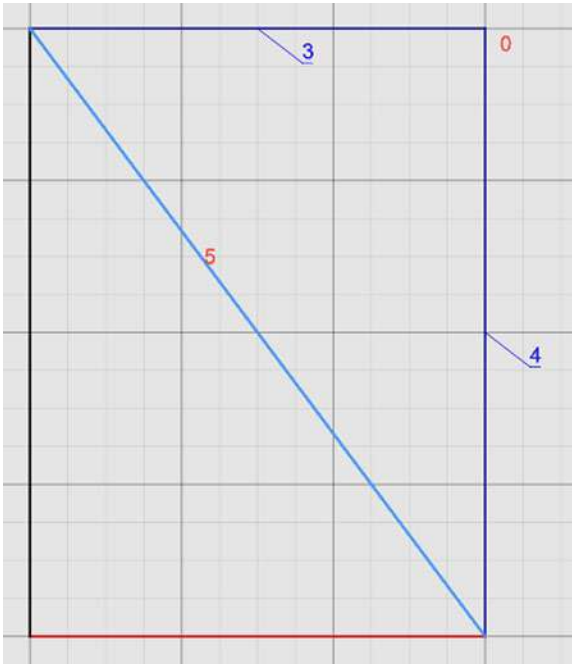
Measure Entities with Ruler



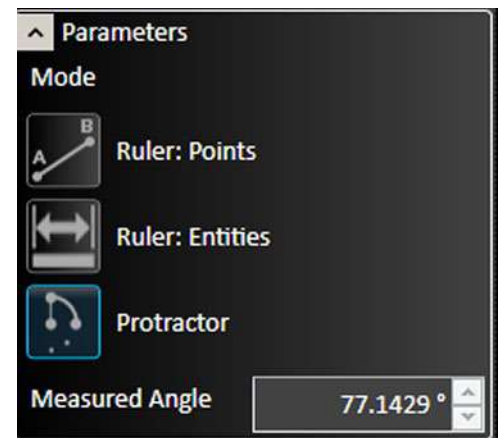
You can measure entities in one of three modes: as a **ruler** between **points** or between **entities**, or as a **protractor** between points on an arc. The tool allows for measurement between any two points, and displays offset in X and in Y directions.



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.



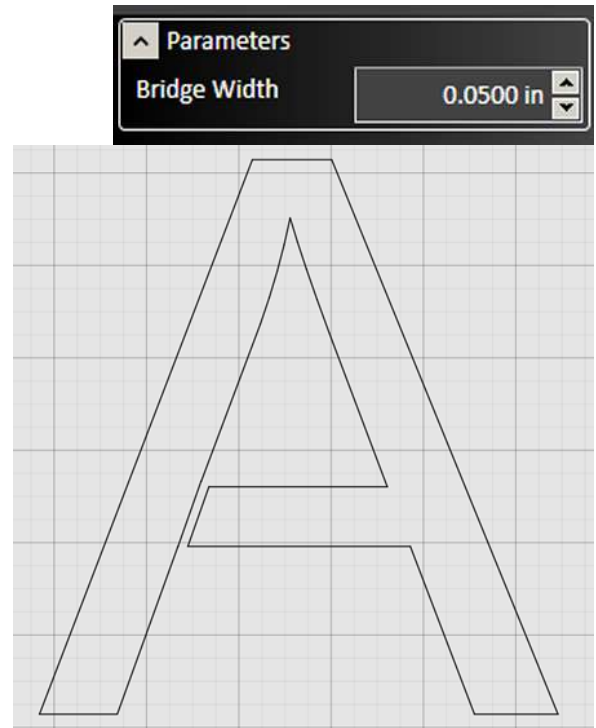
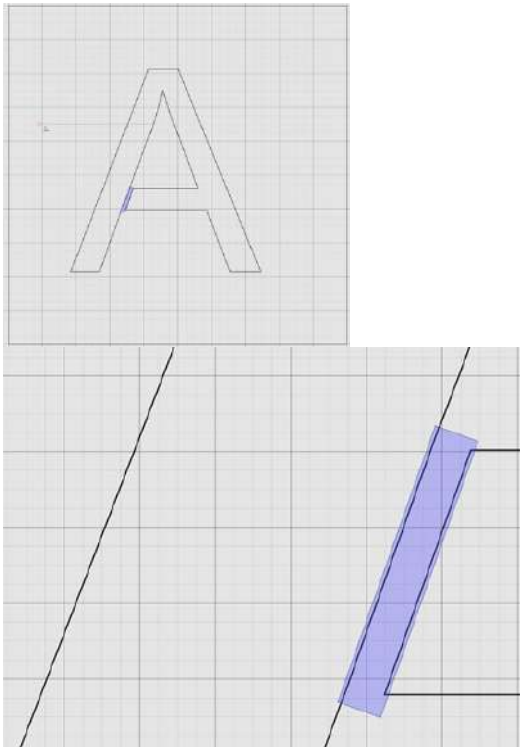
Protractor used to measure the arc and **Measured Angle** between two points of a polygon. Note that the protractor will not measure an arc or angle if it is grouped with other lines or shapes. You must ungroup the arc to enable the protractor tool.

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.

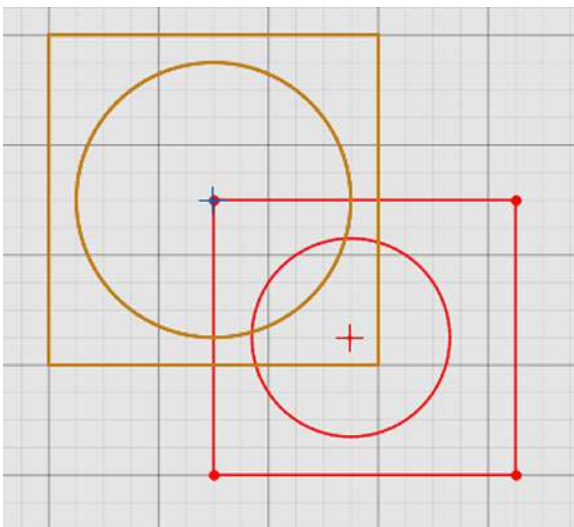
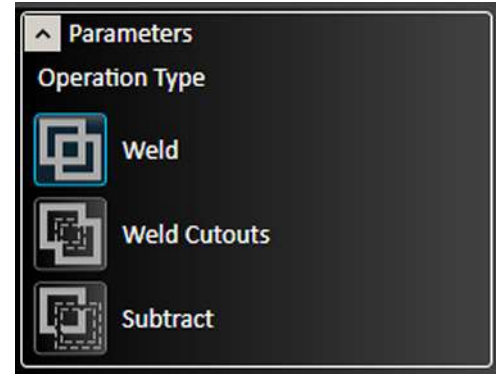
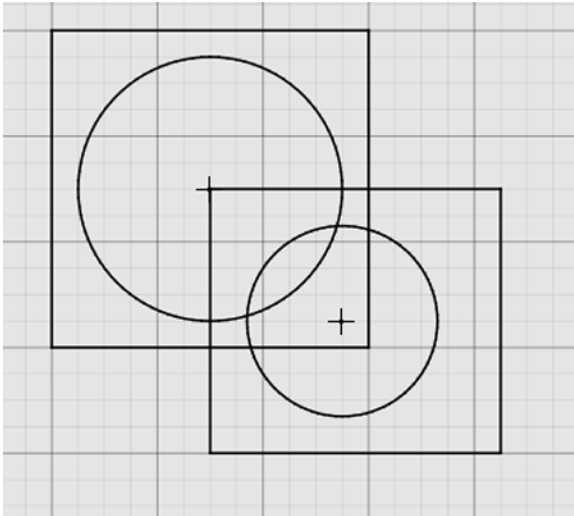


Perform Boolean Operation on Entities

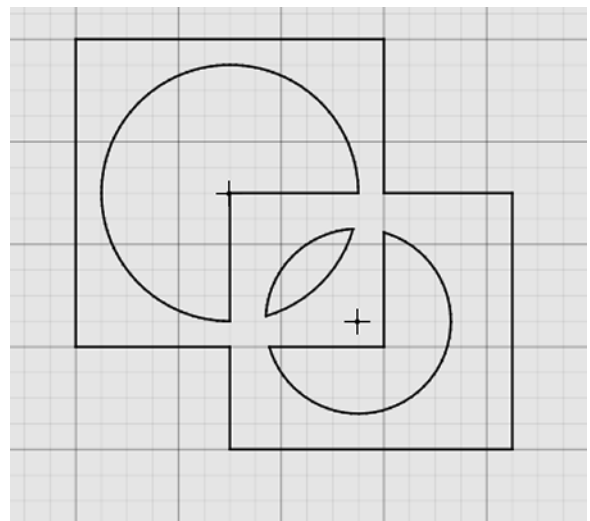
Perform Boolean Operation can be used to modify one shape by adding or subtracting another shape to it. The three available operations are **Weld** (combine) solid areas, **Weld Cutouts**, or **Subtract**.



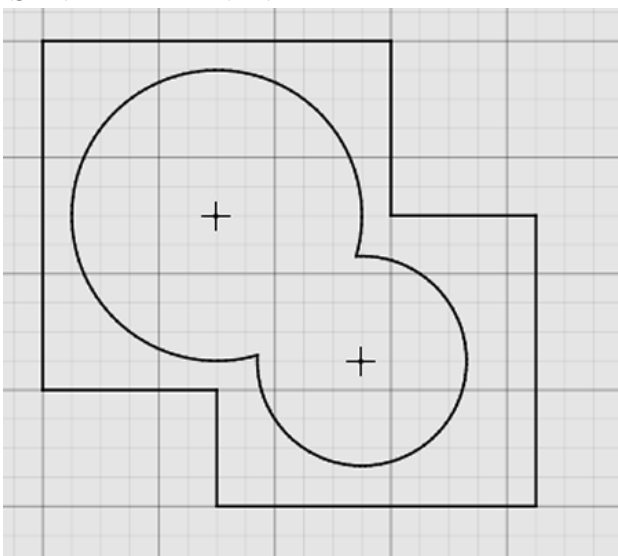
Select Perform Boolean Operation, select the specific operation in the parameters window, then select the objects one at a time, right clicking to confirm each selection. The first object selected will be modified by the second. The example below shows the results of all three operations on the initial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



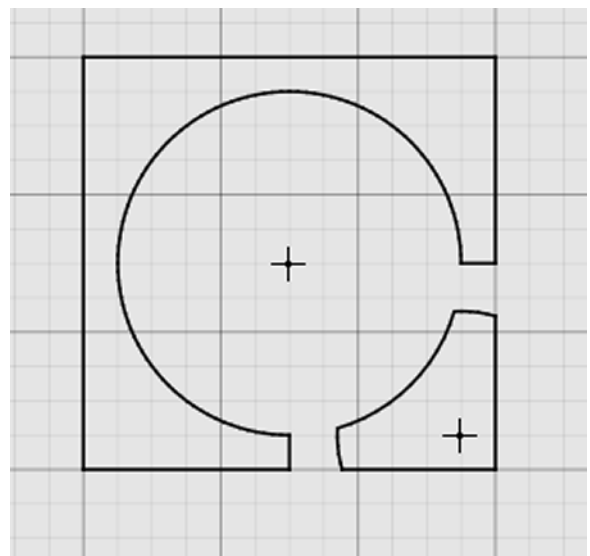
Shapes highlighted to indicate which object is first (gold) and second (red).



The results of a **Weld** operation.



The results of a **Weld Cutouts** operation



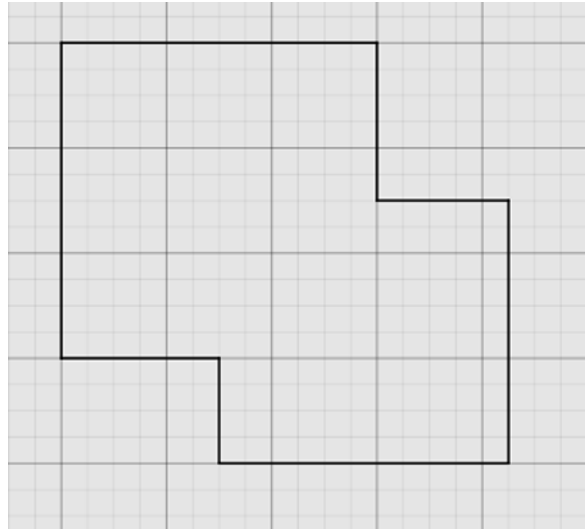
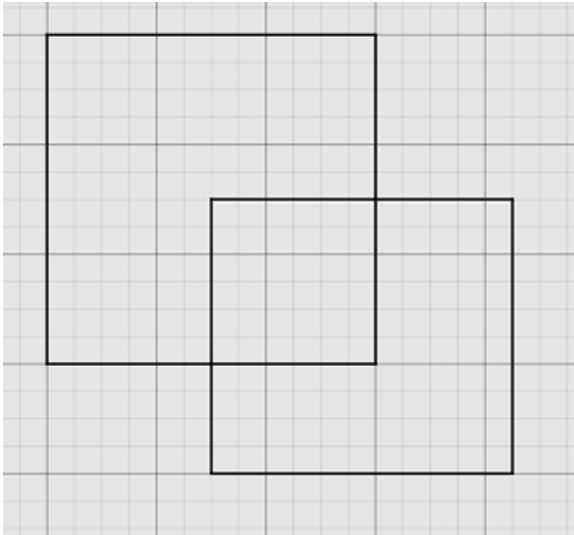
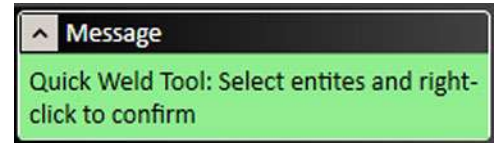
The results of a **Subtract** operation.

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.



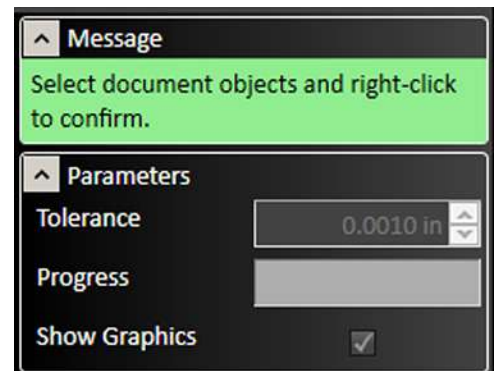
Arc Fitting Tool



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

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

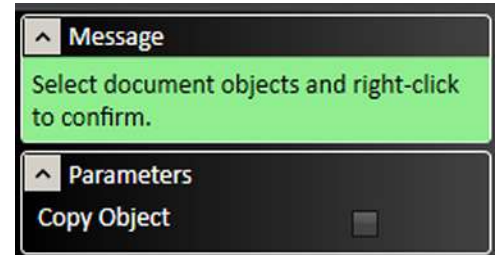
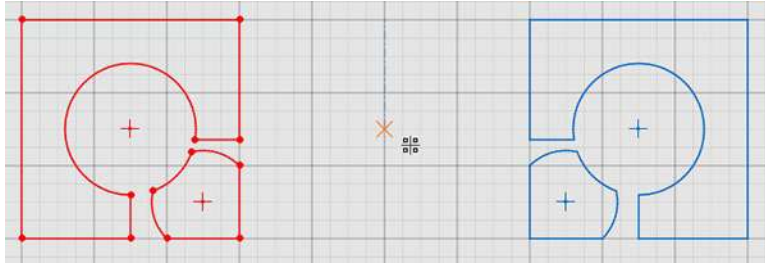
When **Show Graphics** is selected, an outline lines and arcs will be displayed in the drawing window..



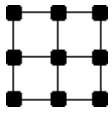
Mirror Entities



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



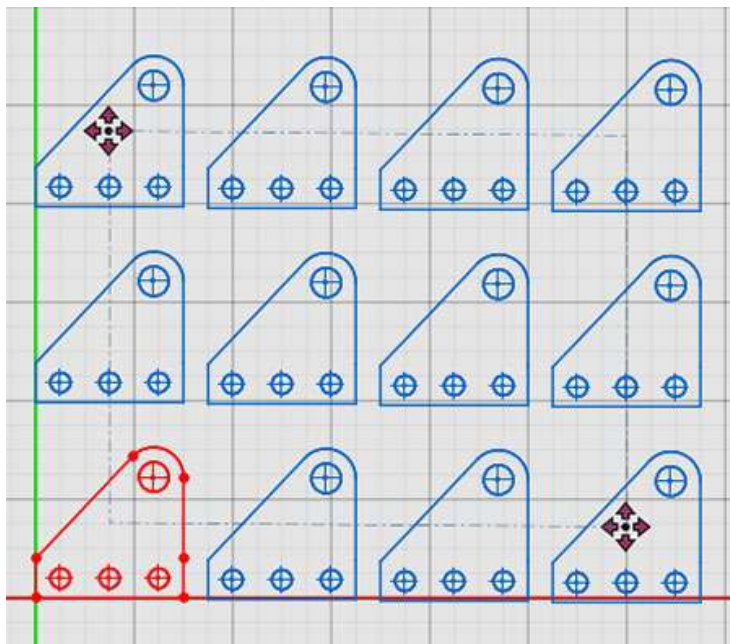
The linear pattern tool duplicates selected features in a grid fashion. Select the feature to be duplicated, select the **Linear pattern** tool then specify parameters.

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.

Direction 1	
Spacing	1.0000 in
Total Distance	0.0000 in
Count	1
Angle	0.0000 °

Direction 2	
Spacing	1.0000 in
Total Distance	0.0000 in
Count	1
Angle	90.0000 °



Direction 1	
Spacing	3.5001 in
Total Distance	10.5004 in
Count	4
Angle	-0.4769 °

Direction 2	
Spacing	3.9709 in
Total Distance	7.9417 in
Count	3
Angle	90.0000 °

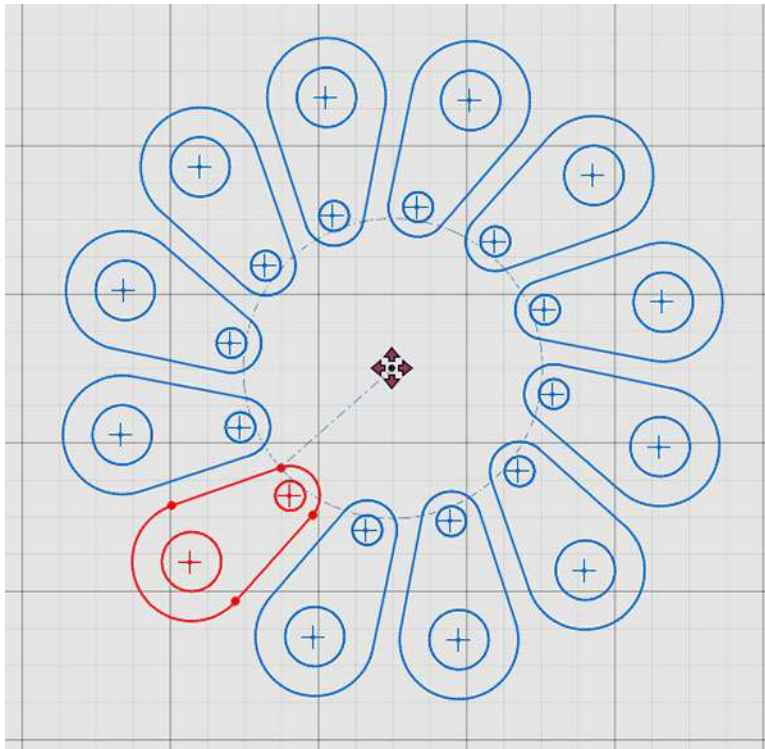
Circular Pattern Tool



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.

Parameters	
Center X	0.0000 in
Center Y	0.0000 in
Spacing Mode	☐
Angle	360.0000 °
Count	1



Parameters	
Center X	22.5000 in
Center Y	17.5000 in
Spacing Mode	☒
Angle	30.0000 °
Count	12

Pan and zoom tools



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



Pan

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



Zoom Extent

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



Zoom

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



Zoom to Fit

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

Snap tools



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

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

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

- **All.snap**
- **Center.snap**
- **Endpoint.snap**
- **Grid.snap**
- **Horizontal/vertical.snap**
- **Intersection.snap**
- **Midpoint.snap**
- **Nearest.snap**
- **Parallel.snap**
- **Perpendicular.snap**
- **Quadrant.snap**
- **Tangent.snap**

All snap



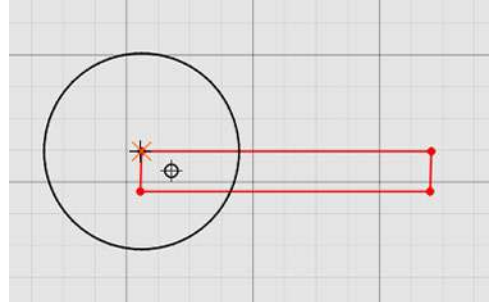
When All snap is selected, individual snap options may be selected.

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

Center snap



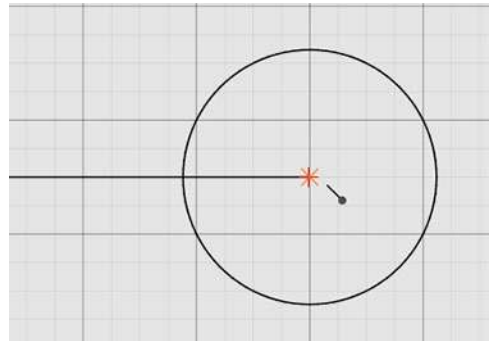
Features connect to the centers of elements.



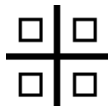
Endpoint snap



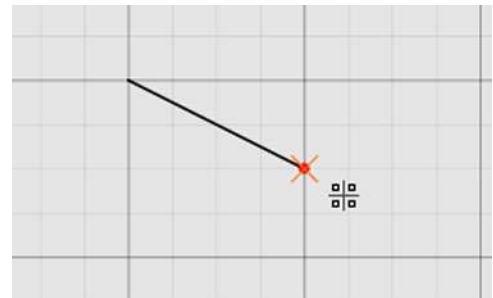
Features connect to the endpoints of elements.



Grid snap



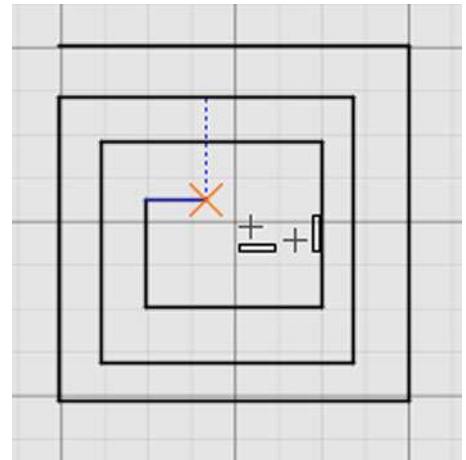
Features connect to points located along the background gridlines.



Horizontal/vertical snap



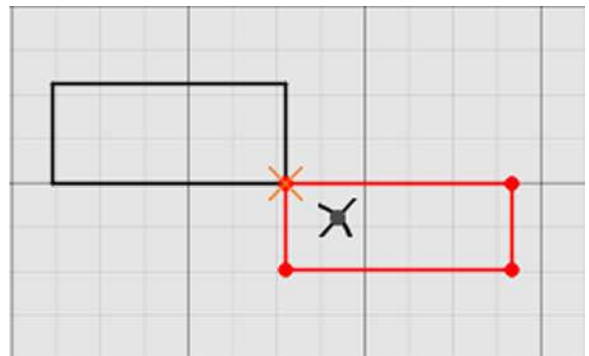
Features align vertically and horizontally when pulled into either direction.



Intersection snap



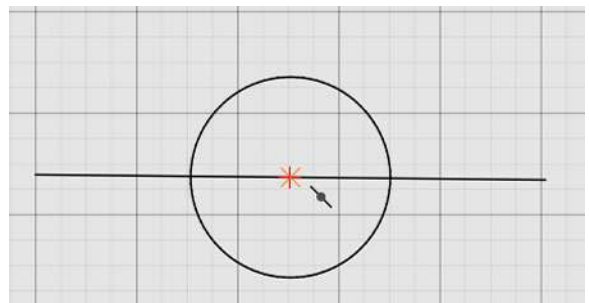
Features connect to the intersection points of elements in the drawing.



Midpoint snap



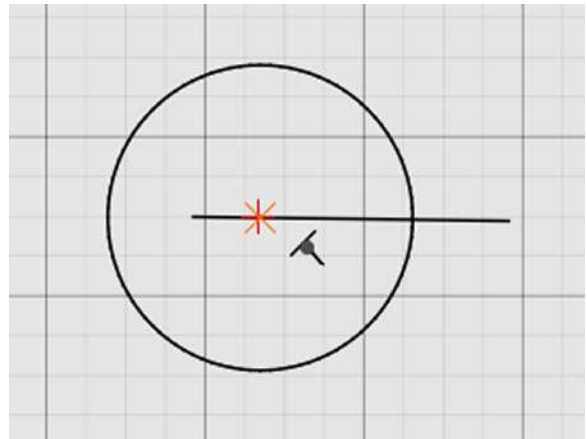
Features connect to the midpoints of elements.



Nearest snap



Features connect to the element closest to the cursor.

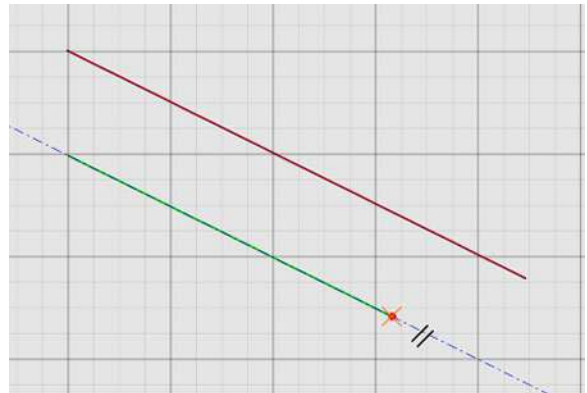


Parallel snap

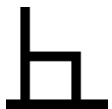


Features snap to a line parallel to the selected element.

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

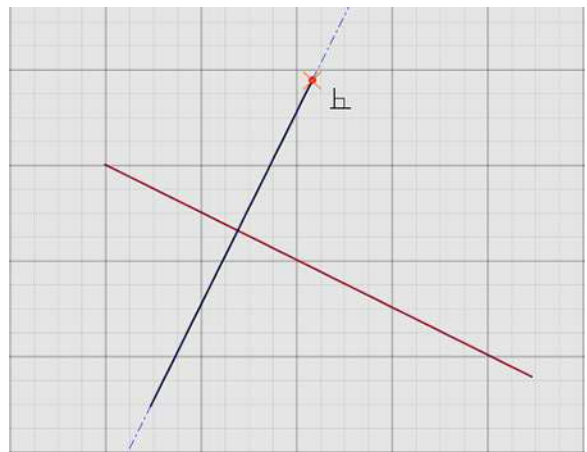


Perpendicular snap



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

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

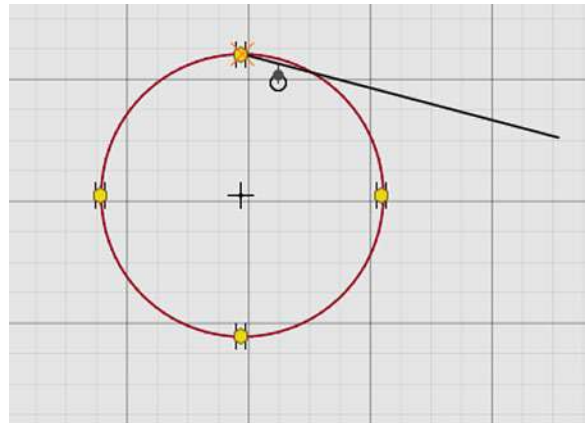


Quadrant snap



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

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

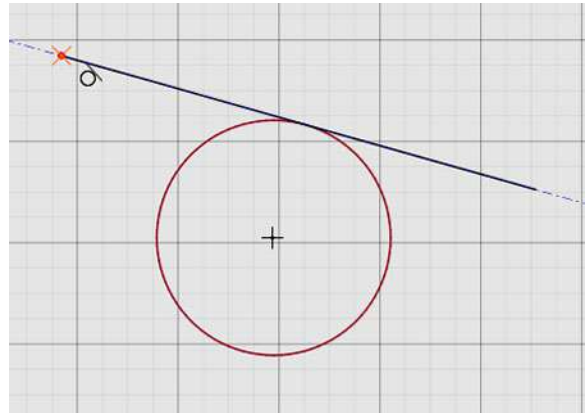


Tangent snap



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.



Free form stretching and manipulation tools

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

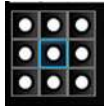
Note that different types of objects respond differently to scaling. For example, circles will expand, and straight line segments will be extended, but ellipses and arcs will be divided into multiple line segments.

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



Control handle grid

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

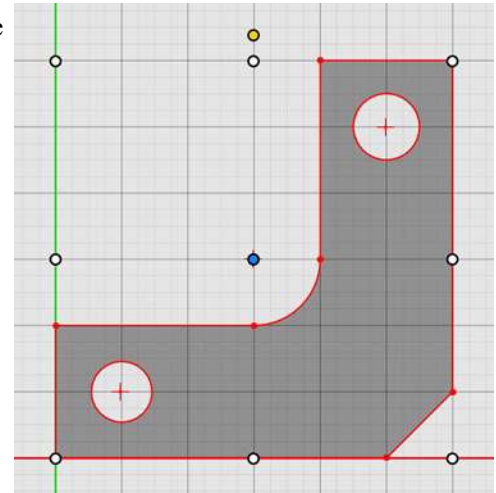


To move the object, click and drag the **blue** control point. The blue point is also used for rotation, and to determine the position of the object.

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.

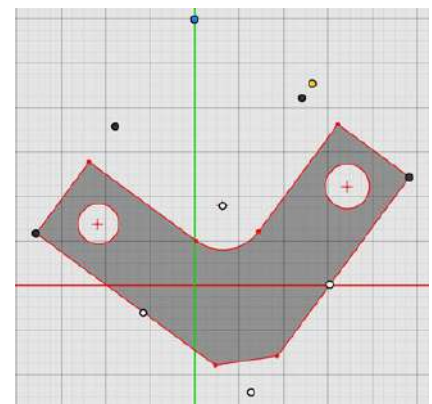
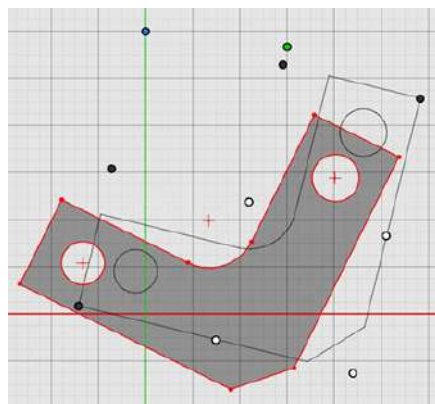
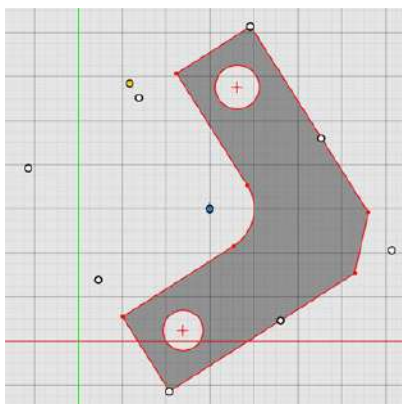
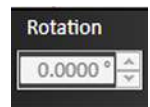


Rotation

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.

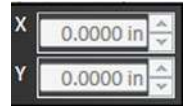
Below, the bracket is shown rotating around the middle (left), and upper right (center, right) control points. Note that the outline of the image is visible (center) during a move to show the new position.



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.

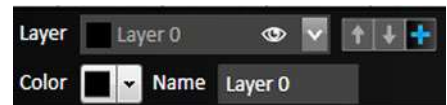
Changes can also be made directly to each field



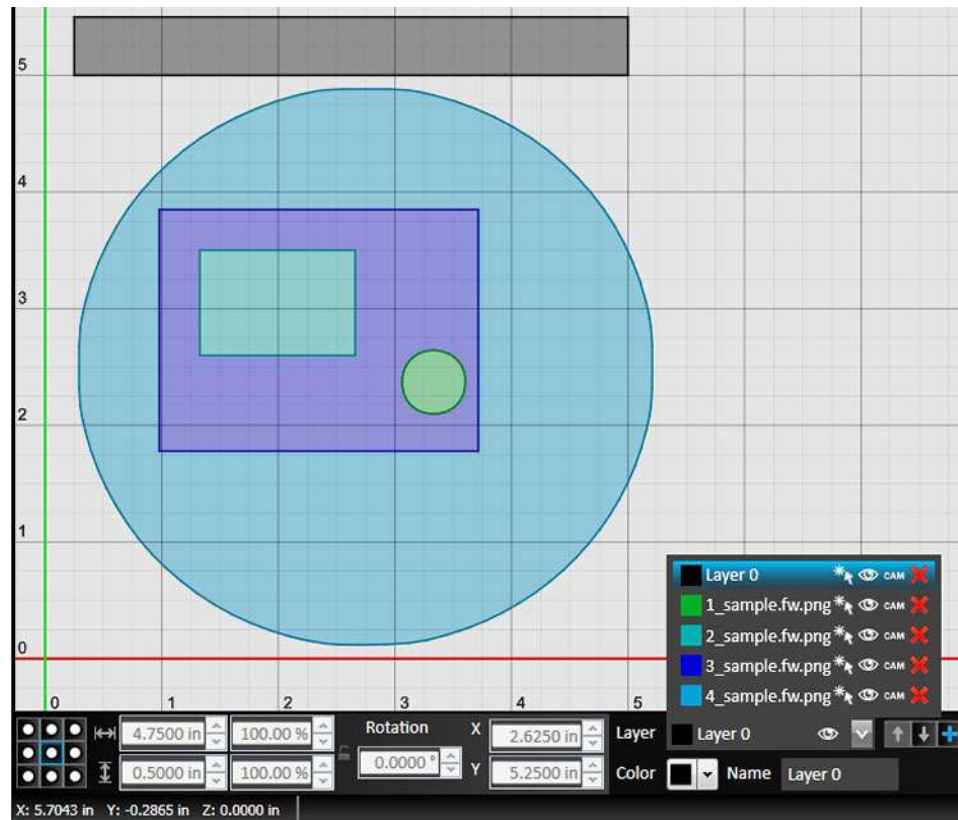
Layers

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 Flash cut CAM.

- **Layer**
- **Color**
- **Name**



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



Layer

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

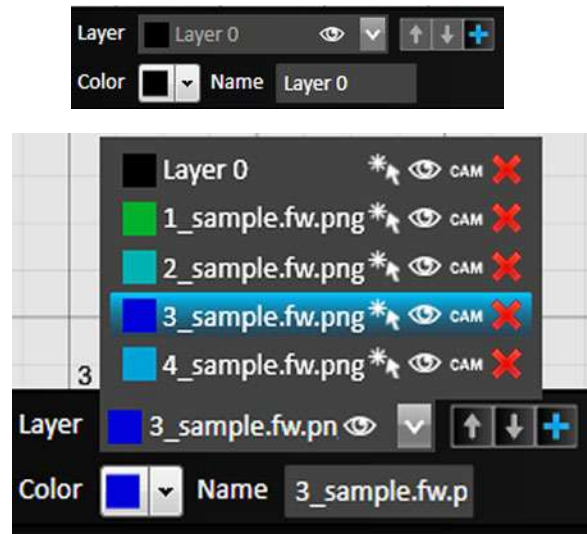
The color, name, and visibility of the active layer are always displayed in the toolbar.

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

Four clickable icons are visible:

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

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



Move layer up/down

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

Add layer

Use the plus sign icon to add a new layer.

Color

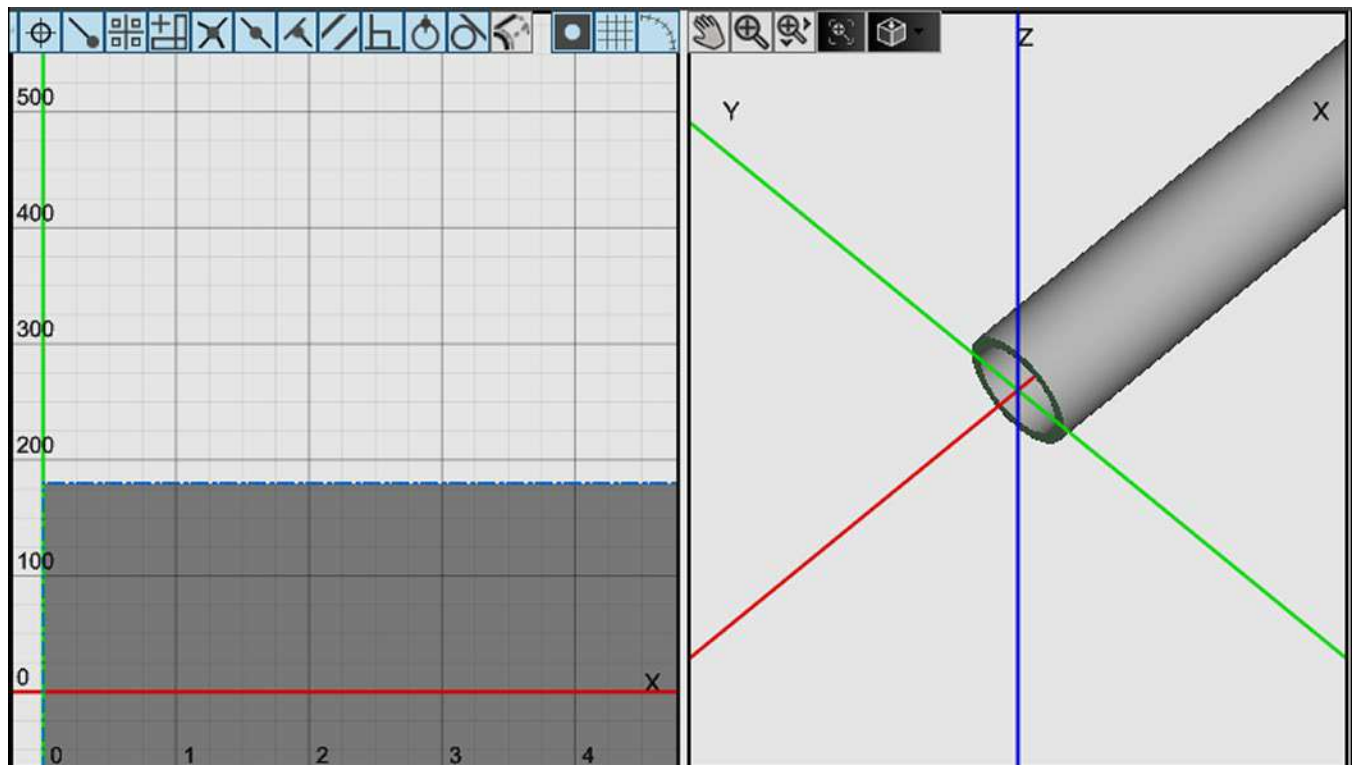
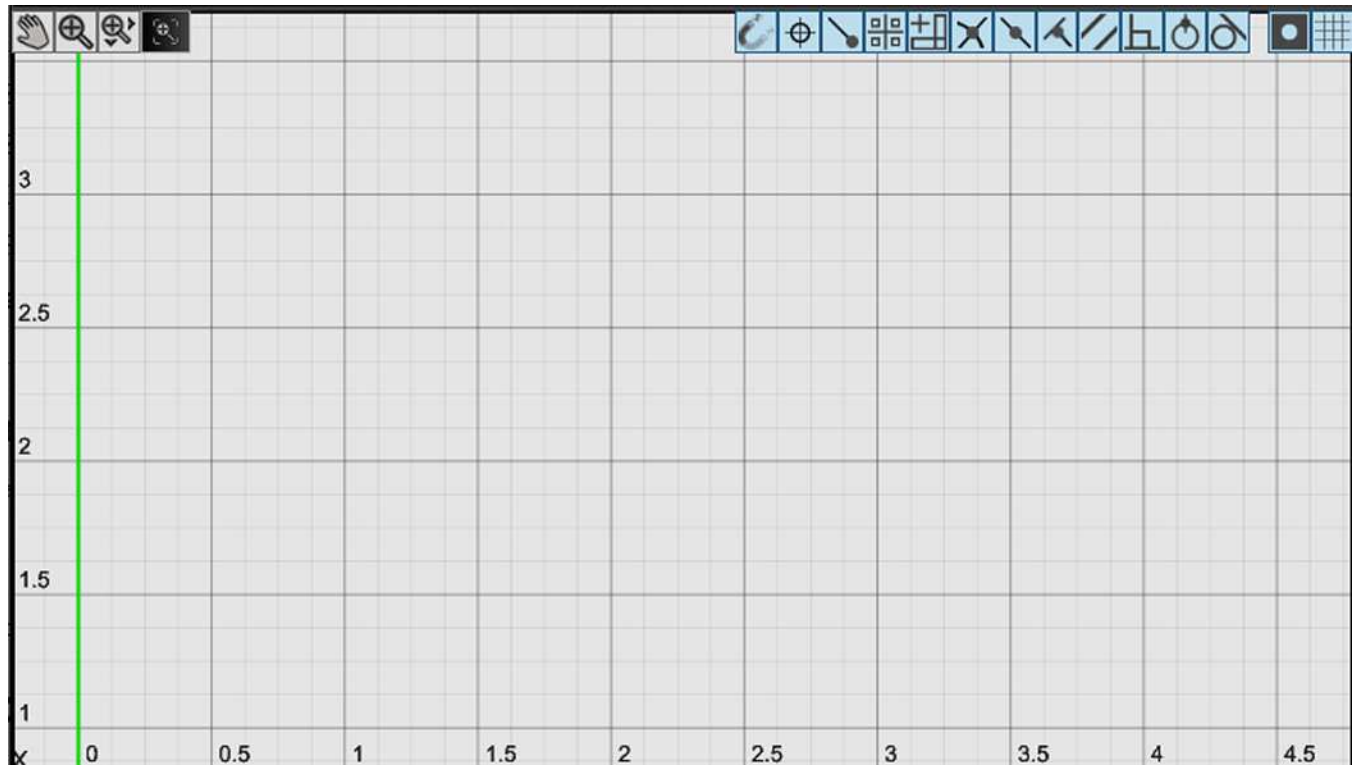
The color of the active layer can be changed here. Clicking the arrow brings up a preset menu of colors. More colors can also be defined, using the Windows color tool.

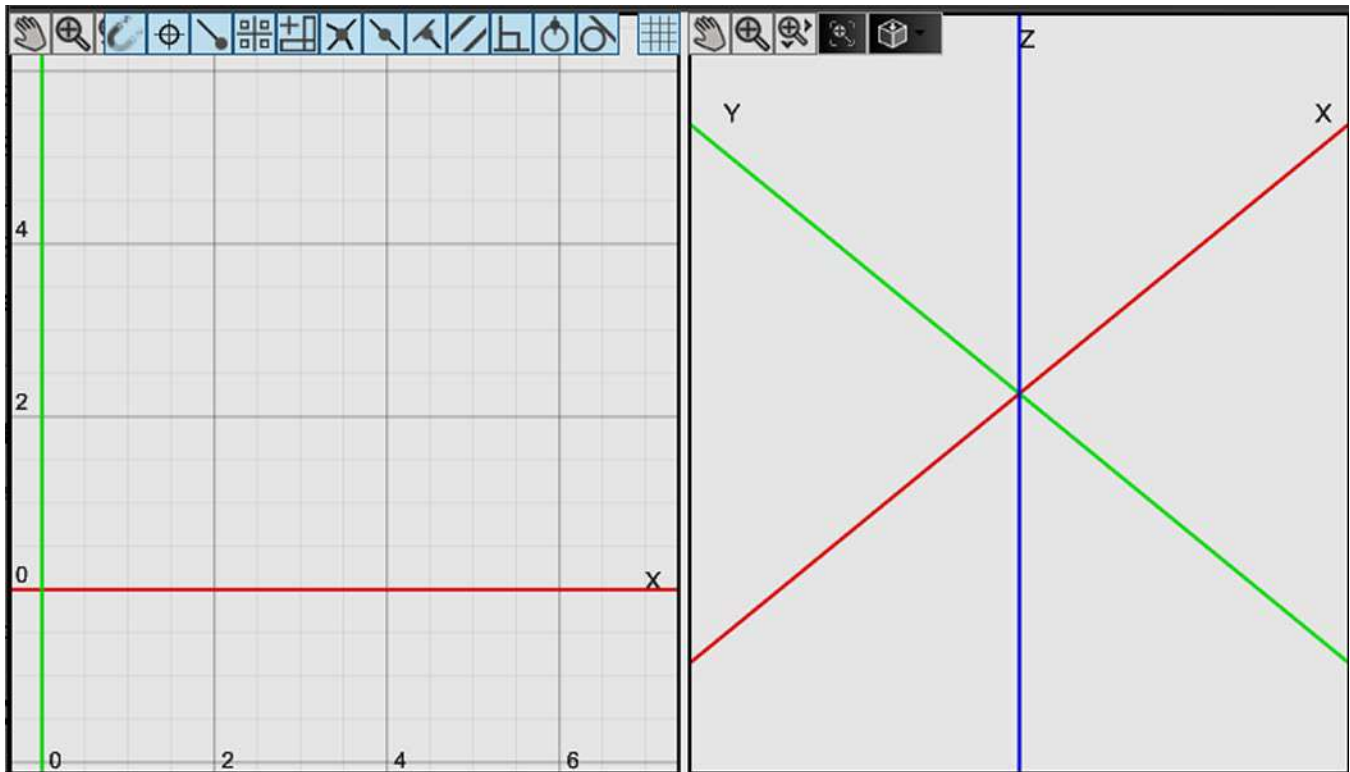
Name

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

Drawing workspace

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





Viewport tools

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

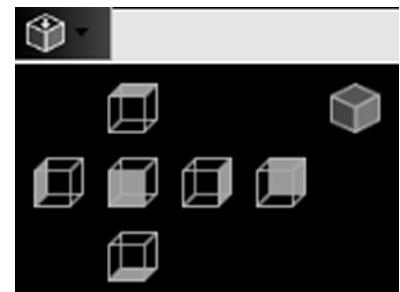
Pan and zoom tools

The **Pan and zoom tools** function in the same way as those in the main drawing workspace.

Orientation tool

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

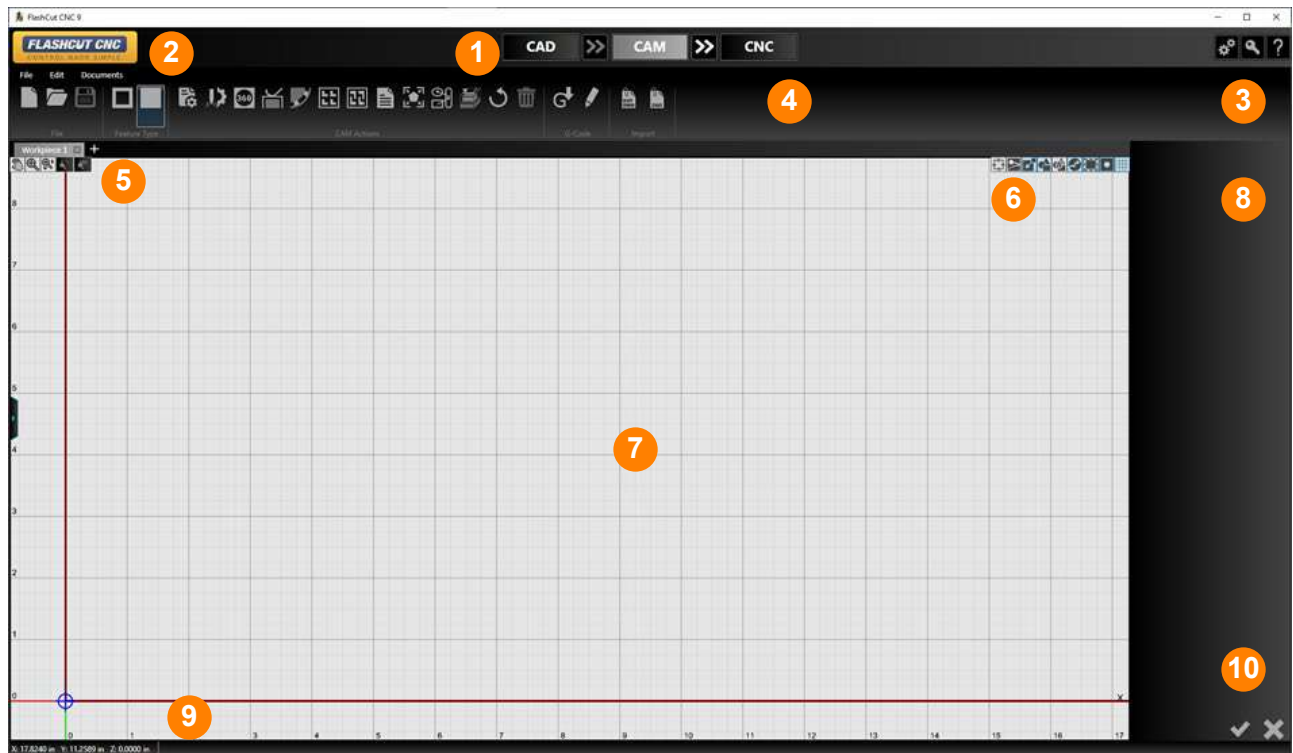
Position the cursor over the orientation tool icon to display and select either a specific face of an orthographic view (front, top, left, etc.), or an isometric view.



FlashCut CAM

FlashCut CAM (Computer-Aided Manufacturing) generates an accurate toolpath from a CAD drawing. This toolpath is used to create the G-Code that will be used in FlashCut CNC.

The FlashCut CAM main screen is shown here. An explanation of each area of the screen follows.



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

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

Menu bar print commands

Print

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

- **Print Labels**
- **Print Nest**

Print Labels

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

Page Setup

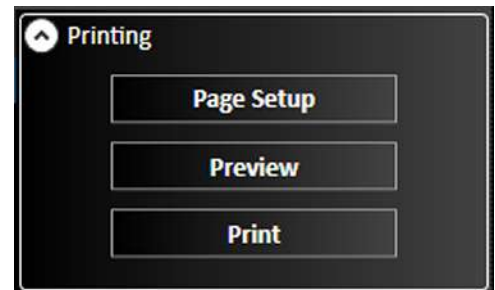
Brings up a Windows page setup window, allowing you to set page size, layout, paper source, and all margins.

Preview

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

Print

Brings up a Window printer selection window.



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 Attributes

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

Horizontal/Vertical Alignment

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

Horizontal/Vertical Margin

Set margins for label text.

Print Nest

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

Page Setup, Preview, Print

Same as for **Print Labels**

Resolution (DPI)

Set value up to 600 DPI. Default is 150 DPI.

Parameters

Label Spacing X	0.0000 in
Label Spacing Y	0.0000 in
Mode	Specify Label Si
Label Height	1.5000 in
Label Width	1.0000 in
Font Name	(TT) Arial
Font Size	12
Bold	☐
Italic	☐
Horizontal Alignment	Center
Horizontal Margin	0.0000 in
Vertical Alignment	Center
Vertical Margin	0.0000 in

Printing

Page Setup	
Preview	
Print	
Resolution (DPI)	150.0000

Print Area Coordinates

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 tools** for definitions.

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

Parameters

Print Area:

From X

0.0000 in

From Y

0.0000 in

To X

48.0000 in

To Y

48.0000 in

Set Whole Workpiece

Scale:

Standard Scale

-

Scale

13.5417 %

Fit To Page

Display Settings

Parts

☒

Part Names

☐

Cutting Directions

☐

Toolpath

☒

Kerf

☐

Program Zero

☒

Shading

☒

Grid

☒

Ribbon



The FlashCut CAM ribbon has six types of command icons:

- **File tools**
- **Feature Type**
- **Selection**
- **CAM Actions**
- **G-Code**
- **Import**

File tools

There are two file tools on the ribbon:

- **Open (Ctrl+O)**
- **Save (Ctrl+S)**

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 5 or later. This command is unavailable when no recent changes have been made.

Feature Type

This pair of settings controls whether the top level of the drawing, indicated by the outermost line, is considered a part or a cut out. This setting will change which sections of the drawing are considered scrap, and which the 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. These options will be greyed out in the ribbon.

Make Top Level Feature a Part



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

Note This option is selected by default.

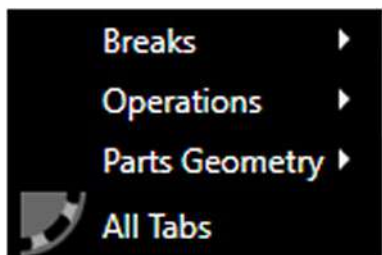
Selection



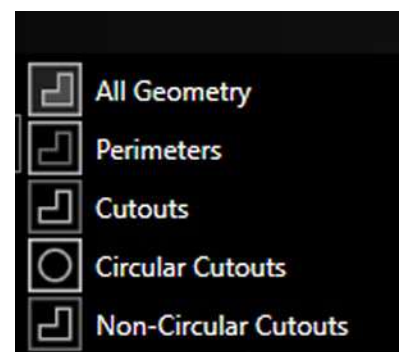
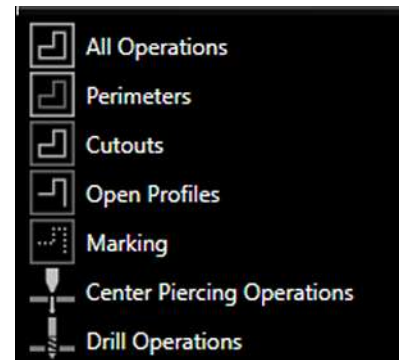
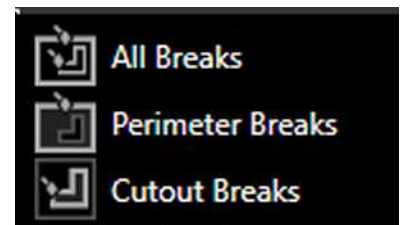
The **Selection** icon is visible if the installed Fab Head supports it. At present, only **Spindle** Fab Heads support Selection.

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

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



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



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.

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

- **Project.Settings**
- **Plasma.Settings**
- **Oxyfuel.Settings**
- **Waterjet.Settings**
- **Laser.Settings**
- **Marker.Settings**
- **Spindle.Settings**
- **Knife.Settings**
- **Change.Project.Break.Settings**
- **Change.Project.Tab.Settings**
- **Grid.Nesting**
- **TrueShape.Nesting**
- **Nesting.Information**
- **Fit.workpiece.to.the.parts**
- **Sequence.tool**
- **Simulation.Tool**
- **CAM.Reset**
- **Delete**



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

Project Settings



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

General

Default Cutting Fabhead

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

Default Marking Fabhead

Select the default marking fabhead.

Workpiece

Width

Specify the width of the sheet being cut. Changes to this value will change the size of the material in the drawing window as well as any nesting settings.

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.

The screenshot shows the 'Project Settings' dialog box with the following sections and values:

- General**
 - Default Cutting Fabhead: Powermax85
 - Default Marking Fabhead: (empty)
- Workpiece (Workpiece 1)**
 - Width: 120.0000 in
 - Length: 60.0000 in
 - Recent Sheet Sizes: (empty)
- Material**
 - Material Type: Stainless Steel
 - Material Subtype: *Generic*
 - Thickness: 0.0100 in
 - Recent Materials: (empty)
- Collision Avoidance**
 - Enable Collision Avoidance: ☒
 - Extra Margin: 0.0000 in
 - Max Part Size to Avoid: Specify X and Y
 - X: 10.0000 in
 - Y: 10.0000 in

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.

Collision Avoidance

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

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

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,

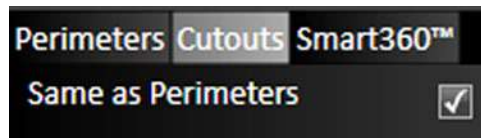
Perimeters

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.

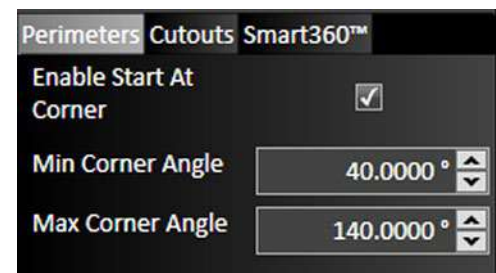
Enable Start At Corner

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.

Leads for Non-Corners	
Overburn	0.0500 in
Turn Off Fabhead Early	Toolpath Compli
Lead In	Arc
Radius	0.2500 in
Length Type	Based on Sheet
Factor	0.0000
Maximum Arc Angle	0.0000 °
Lead Out	Line
Angle	0.0000 °
Length Type	Fixed
Length	0.0000 in

Change Project Tab Settings



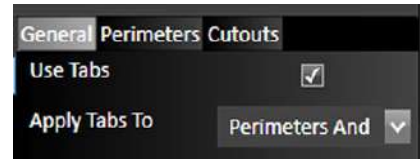
Tabs are used to keep the part attached to the workpiece.

Use Tabs

When selected, tabs will be included in the toolpath for the drawing

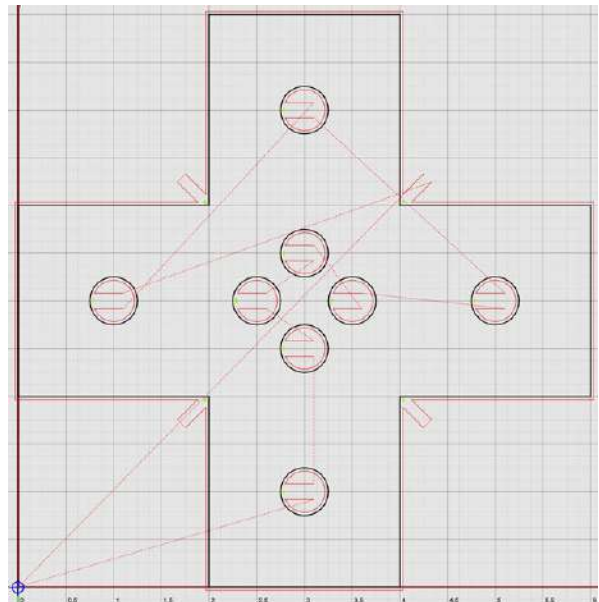
Apply Tabs

Tab settings can be configured for perimeters and cutouts, either separately or together. Three labeled tabs enable detailed configuration.



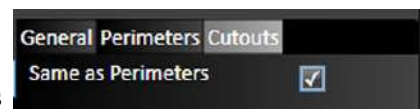
In this drawing, four perimeter tabs have been used to keep the part connected to the workpiece. The tabs appear at the right-angle corners formed by the arms of the cross.

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.



Stitch Tab

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

First Tab Shape Tab Width

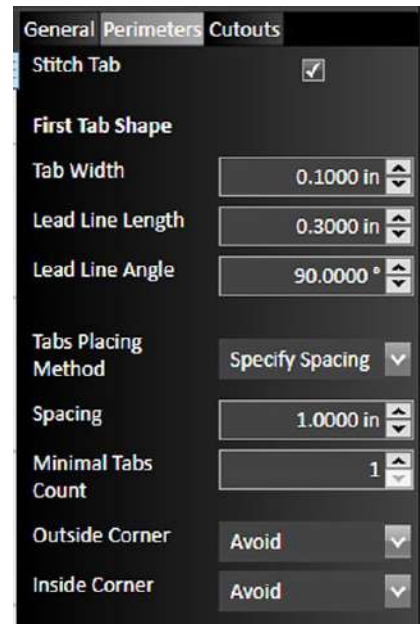
Define tab width

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.

Plasma Settings



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

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various plasma torches. Click the icon to bring up options in the parameters window.

You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either **Accept** or **Cancel** the changes.

The values in the parameters are derived from the settings in the *.setup file. Changes made here will apply to the specific drawing, but will not change the values set in the configuration window. The configured values for the Fab Head are in turn populated from the cut charts. See **Configuring FlashCut** for more details.

Nozzle

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

Note that each of the following parameters will be automatically adjusted when the nozzle is specified. However, they can also be input manually.

Quality Level

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

Lookup/Reset to Default

Return any changed fields to values from cut charts.

Feedrate

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

Kerf Width

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

Powermax85 (Cut)	
Nozzle	85A Shielded
Plasma Gas	Air
Quality Level	Production
Lookup/Reset to Default	
Feedrate (inch/min)	336.0000
Kerf Width	0.0680 in
Pierce Height	0.1500 in
Cut Height	0.0600 in
Pierce Delay	0.0000 sec
Voltage	121.0000 V
Amperage	85.0000 A
Pressure (psi)	85
EdgeStart	☐
Corners Shape	Sharp
Enable Smart360™	☒
Hole Size ≤ Sheet Thickness Factor	3.0000
Reduce Feedrate to	14.0000 %
Additional Kerf	0.0107 in
Suspend THC for Holes	☒

Pierce Height

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

Cut Height

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

Pierce Delay

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

Voltage

This is the target voltage for the THC system in order to maintain a specified cut height. The set point value is determined by the cut chart for each plasma torch, and can also be found in the operator's manual of the plasma torch. It and is dependent on the material type, thickness, torch settings, and other parameters.

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

Amperage

Specifies the operating amperage of the torch.

Pressure

Specifies the air pressure of the torch in psi.

Edge Start

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

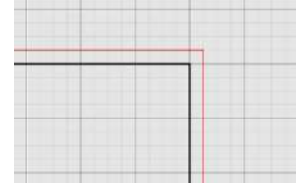
Corners Shape

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.

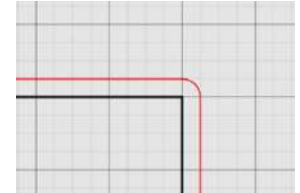
- **Sharp corners**

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



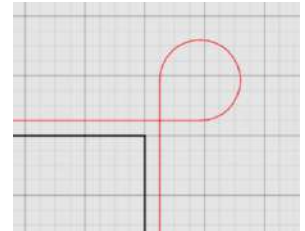
- **Rounded corners**

The toolpath describes an arc at the corner of the part, maintaining a consistent distance from the edge.



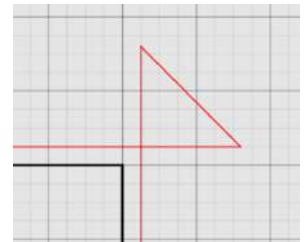
- **Rounded loops**

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.



Enable Smart360™



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

- **Hole Size $\leq$ Sheet Thickness Factor**
- **Reduce Feedrate**
- **Additional kerf**
- **Suspend THC for Holes**

Enable Smart360™	
Hole Size $\leq$ Sheet Thickness Factor	3.0000
Reduce Feedrate to	14.0000 %
Additional Kerf	0.0107 in
Suspend THC for Holes	☒

Oxyfuel Settings



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

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various Fab Heads. Click the icon to bring up options in the parameters window.

You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either **Accept** or **Cancel** the changes.

The values in the parameters are derived from the settings in the *.setup file. Changes made here will apply to the specific drawing, but will not change the values set in the configuration window. The configured values for the Fab Head are in turn populated from the cut charts. See **Configuring FlashCut** for more details.

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.

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.

Apply Small Holes Processing

When selected, enables cleaner cutting of small holes.

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

Lookup/Reset to Default

Return any changed fields to values from cut charts.

Waterjet Settings



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

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various Fab Heads. Click the icon to bring up options in the parameters window.

You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either **Accept** or **Cancel** the changes.

The values in the parameters are derived from the settings in the *.setup file. Changes made here will apply to the specific drawing, but will not change the values set in the configuration window. The configured values for the Fab Head are in turn populated from the cut charts. See **Configuring FlashCut** for more details.

Machinability

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

Nozzle Diameter

Nozzle diameter for the cutting head.

Orifice Diameter

Orifice diameter for the cutting head

Abrasive Flow

Flow rate for the abrasive compound.

Abrasive Factor

Abrasiveness of the compound.

Cutting Quality

Low, fair, average, good, or excellent

Waterjet (Cut)	
Machinability	80.0000
Nozzle Diameter	0.0400 in
Orifice Diameter	0.0150 in
Abrasive Flow	1.5000
Abrasive Factor	1.0000
Cutting Quality	Good
Kerf Width	0.0400 in
Feedrate (inch/min)	72.5250
Pierce Delay	0.0020 sec
Pierce Style	Straight
Lookup/Reset to Default	
Corners Shape	Sharp
Apply Small Holes Processing	☒
Hole Size ≤ Sheet Thickness Factor	1.5000
Reduce Feedrate to	14.0000 %

Kerf Width

Input the width of the kerf of the waterjet. 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.

Pierce Delay

Delay in seconds before a pierce operation.

Pierce Style

Options are **Straight**, **Wiggle**, or **Circular**.

Lookup/Reset to Default

Return any changed fields to values from cut charts.

Lookup/Reset to Default

Return any changed fields to values from cut charts.

Corners

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

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

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

Apply Small Holes Processing

When selected, enables cleaner cutting of small holes.

- **Hole Size $\leq$ Sheet Thickness Factor**
- **Reduce Feedrate to n%**

Laser Settings



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

FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various Fab Heads. Click the icon to bring up options in the parameters window.

You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either **Accept** or **Cancel** the changes.

The values in the parameters are derived from the settings in the *.setup file. Changes made here will apply to the specific drawing, but will not change the values set in the configuration window. The configured values for the Fab Head are in turn populated from the cut charts. See [Configuring.FlashCut](#) for more details.

Gas

Type of gas used in the cutting head.

Nozzle Diameter

Nozzle diameter for the cutting head.

Lookup/Reset to Default

Return any changed fields to values from cut charts.

Feedrate

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

Kerf Width

Input the width of the kerf of the laser's path. 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

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

Focal Length

Focusing distance of cutting head.

Laser (Cut)	
Gas	Air v
Nozzle Diameter	1.5 v
Lookup/Reset to Default	
Feedrate (inch/min)	35.0000 up down
Kerf Width	0.0500 in up down
Cut Height	0.0600 in up down
Pierce Height	0.1250 in up down
Focal Length	0.0000 up down
Pierce Delay	0.5000 sec up down
End of Cut Delay	1.0000 sec up down
Power Level (watts)	1000.0000 up down
Pressure (bar)	85 up down
Corners Shape	Sharp v
Apply Small Holes Processing	☒
Hole Size ≤ Sheet Thickness Factor	1.5000 up down
Reduce Feedrate to	14.0000 % up down

Pierce Delay

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

End of Cut Delay

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

Power Level

Power level (watts) of the laser cutting head.

Pressure

Pressure of the Fab Head's assist gas jet

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.

Apply Small Holes Processing

When selected, enables cleaner cutting of small holes.

- **Hole Size $\leq$ Sheet Thickness Factor**
- **Reduce Feedrate to n%**

Marker Settings



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



See [Configuring FlashCut](#) for more details.

Spindle Settings



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

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.

Spindle (3D Cut)	
Clearance Height	2.0000 in
Plunge Start Height	0.2000 in
Peck Retract Distance	0.0500 in

Knife Settings



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

Feedrate

Rate when cutting.

Plunge Rate/Retract Rate

Rate during negative lift axis movements.

Lift Height

Height of cutting head above workpiece during cuts.

Safe Height

Height of cutter during rapid moves.

Maximum Rotation while Engaged

Degrees of rotation permitted.

Corner Looping

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

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

Corner Undercut Tolerance

Set value.

Minimum Cutting Radius

Set value.

Overcut Depth

Distance of cut through workpiece.

Knife (Cut)	
Feedrate (inch/min)	100.0000
Plunge Rate (inch/min)	100.0000
Retract Rate (inch/min)	100.0000
Lift Height	0.2500 in
Safe Height	1.0000 in
Maximum Rotation while Engaged	15.0000 °
Corner Looping	☒
Looping	Rounded Loops
Extension Distance	0.0000 in
Maximum Angle to Add Loop	150.0000 °
Corner Undercut Tolerance	0.0200
Minimum Cutting Radius	0.1250 in
Overcut Depth	0.0000 in
Open End/Inside Corner Handling	
Single Sided Cutting Knives	Undercut (Incon)
Dual Sided Cutting Knives	Undercut (Incon)
Knife Rotation Smoothing	
Tangential Tolerance	5.0000 °
Max Transition Distance	0.0600 in

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

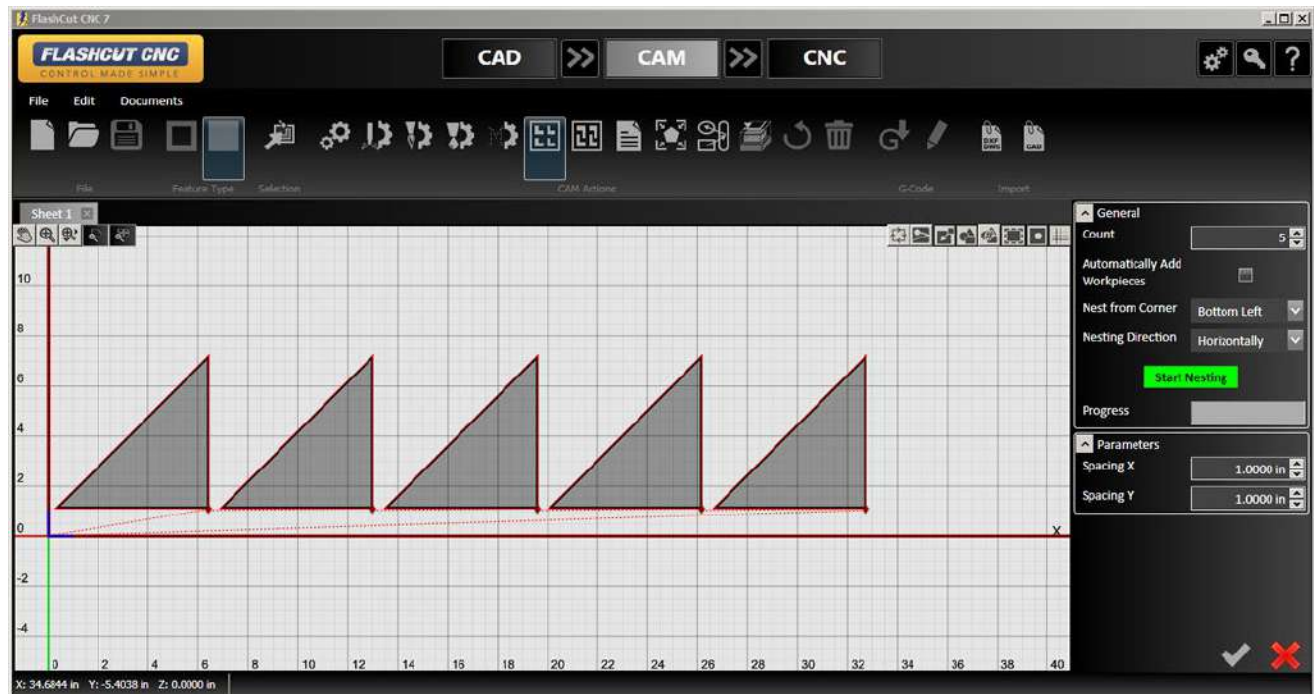
- **Tangential Tolerance**
 - **Max Transition Distance**
-

Grid Nesting



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

Copies fill rows left to right, and when the edge of the material is reached, they will advance up a row. Grid nesting does not change the orientation of any of the parts being nested as illustrated in the screen below where 5 triangles are grid nested. The parameters for grid nesting are described below.



Count

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

Automatically add workpieces

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

Nest from corner

Selects the starting point of the nesting operation.

Nesting direction

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

Start Nesting

Starts the nesting operation

Progress

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

Spacing X

Sets the horizontal spacing between parts.

Spacing Y

Sets the vertical spacing between parts.

The screenshot displays the FlashCut software's nesting configuration window. It is divided into two main sections: 'General' and 'Parameters'. The 'General' section includes a 'Count' field set to 30, an 'Automatically Add Workpieces' checkbox (unchecked), a 'Nest from Corner' dropdown menu set to 'Bottom Left', and a 'Nesting Direction' dropdown menu set to 'Horizontally'. A prominent green 'Start Nesting' button is located below these options. A 'Progress' bar is visible at the bottom of the 'General' section. The 'Parameters' section, located below 'General', contains two fields: 'Spacing X' and 'Spacing Y', both set to '1.0000 in'.

General	
Count	30
Automatically Add Workpieces	☐
Nest from Corner	Bottom Left
Nesting Direction	Horizontally
Start Nesting	
Progress	

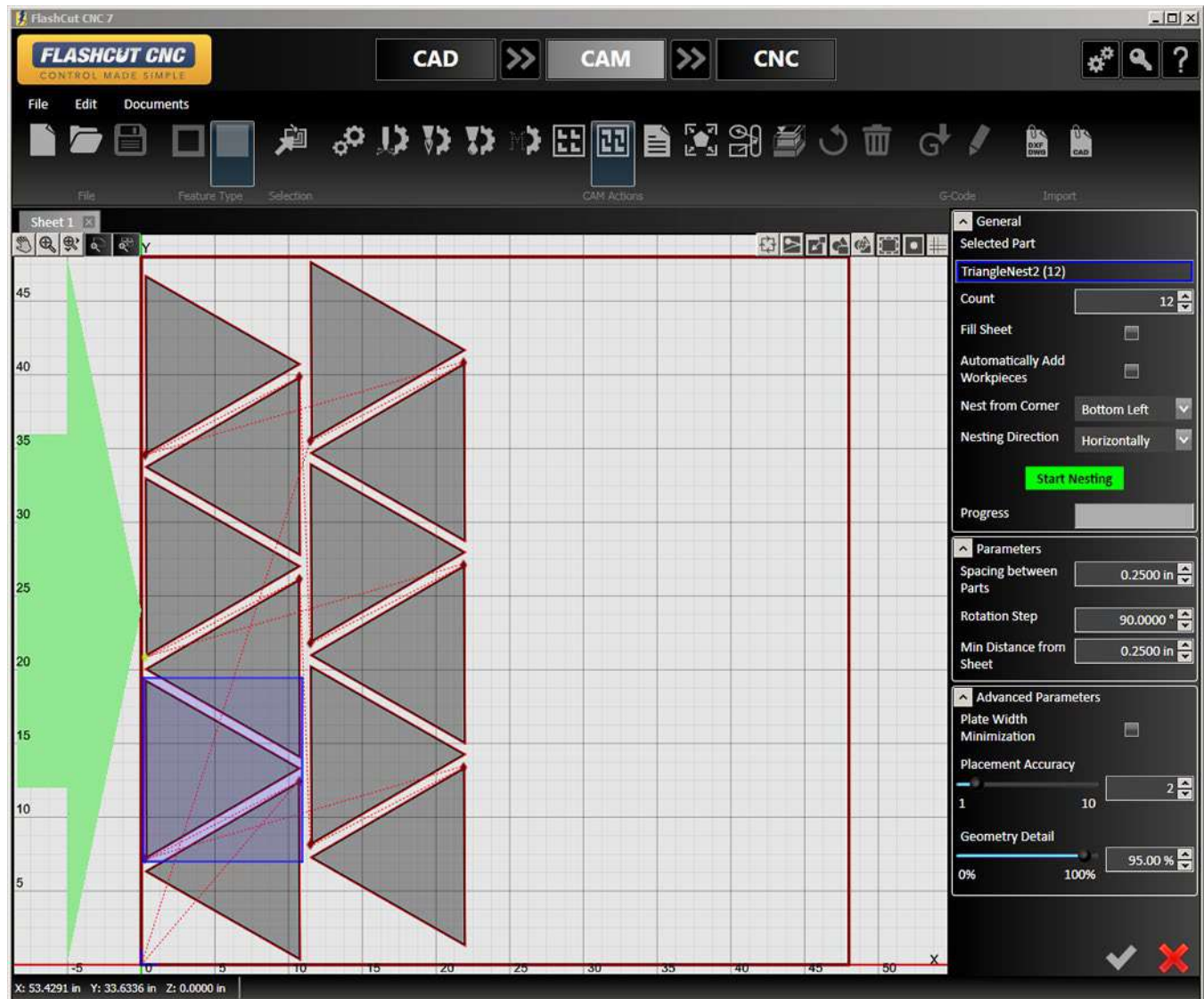
Parameters	
Spacing X	1.0000 in
Spacing Y	1.0000 in

TrueShape Nesting



The TrueShape nesting function will duplicate, rotate, and translate each part to create a nest that minimizes the amount of material used in a sheet.

Twelve triangles are nested below using TrueShape Nesting. Notice the material usage is much lower for TrueShape nesting as it is for the same number of triangles using Grid nesting. The parameters for TrueShape Nesting are described below.



Selected Part

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

Count

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

Fill Sheet

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

Automatically add workpieces

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

Nest from corner

Selects the starting point of the nesting operation.

Nesting direction

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

Start Nesting

Starts the nesting operation

Progress

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

Spacing between Parts

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

General

Selected Part
TriangleNest2 (12)

Count
12

Fill Sheet
☐

Automatically Add Workpieces
☐

Nest from Corner
Bottom Left

Nesting Direction
Horizontally

Start Nesting

Progress

Parameters

Spacing between Parts
0.2500 in

Rotation Step
90.0000 °

Min Distance from Sheet
0.2500 in

Advanced Parameters

Plate Width Minimization
☐

Placement Accuracy
2

Geometry Detail
95.00 %

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%.

Nesting Information



Generates a report from the nested parts, estimating material use, waste, and cost.

A sample generated from the nested triangle example (above) appears here (right). Note that material price can be input directly into the field. All other information is generated automatically from the drawing.

^ Parameters

Nesting Info:

Material Price (per lb) 0.0000

Job Name: TriangleNest2

Sheet No: Sheet 1

Parts List:

TriangleNest2(12)

Sheet Size: 48 x 48 in

Material: Mild Steel

Thickness: 0.0598 in

Total Cut Length: 439.343 in

Total Pierce Count: 12

Total Part Area: 748.24 in^2

Scrap Area: 1516.65 in^2

Scrap Weight: 25.721 lb

Scrap Percentage: 65.82 %

Kerf Area: 39.1 in^2

Scrap Cost: 0

Fit workpiece to the parts



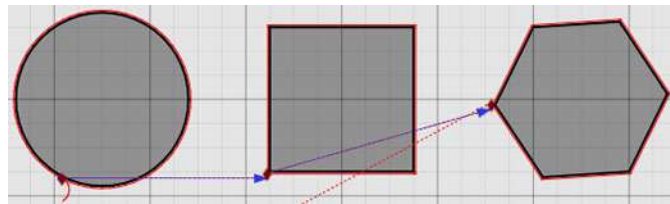
Shrinks the workpiece to fit the area defined by the parts.

Sequence tool



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

To change the order, click the edge of a feature and drag the arrow to the next feature you want to execute. When finished, select the green check mark to finalize changes.



Part sequence mode

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

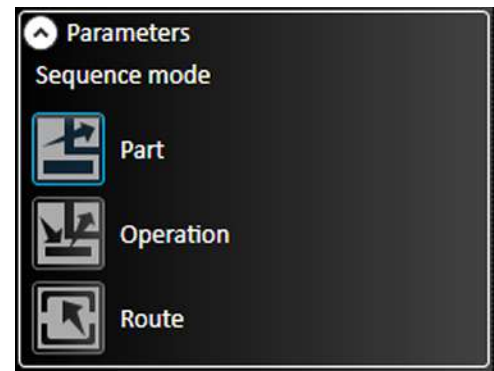
Operation sequence mode

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



Simulation Tool

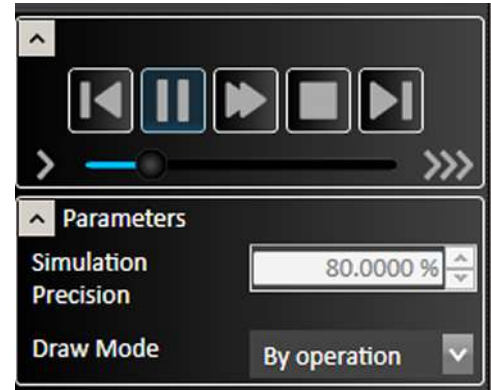


Allows visualization and analysis of the material removal process.

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

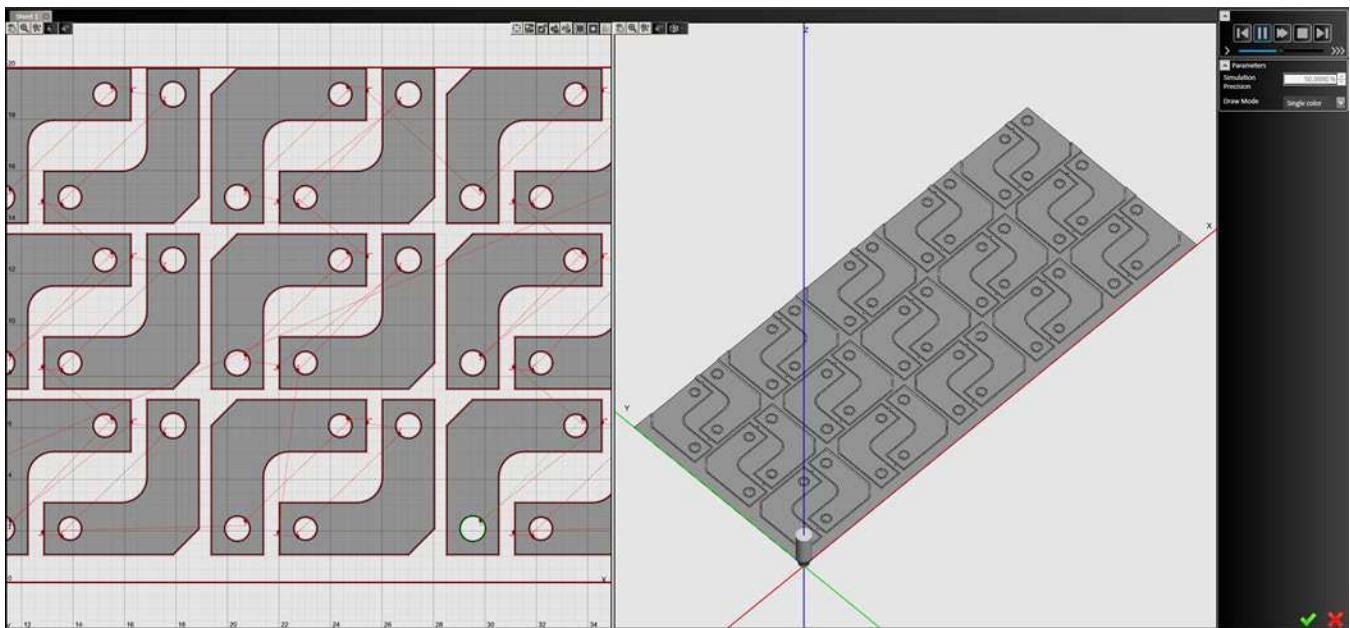


Simulation Precision

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**
- **Open editor**



Create G-Code file



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

Open editor



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

Import

Files can be imported into the CAM drawing workspace. If other objects are added, new toolpaths will be generated.

Note Objects imported as files will not appear in the CAD drawing space.

DXF/DWG



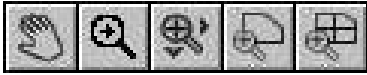
Imports DXF/DWG file to the CAM drawing workspace. The file selection dialog, and other options are shown in the **Parameters** window.

CADCAM



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

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 you to focus in on a specific section of the diagram.



Orientation tool

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

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



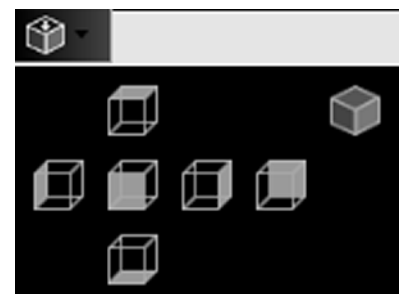
Zoom Workpiece Extents

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.



Display options tools



The display options tools are shown at the top of the drawing workspace and allow you to decide what appears in the drawing window. Click the buttons to toggle each option on and off. You may toggle any number of options at one time.

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. In order 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.

Drawing workspace

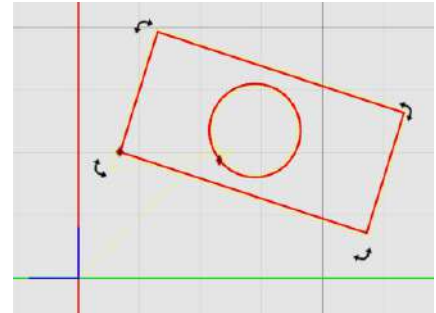
This section discusses the following topics:

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

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.

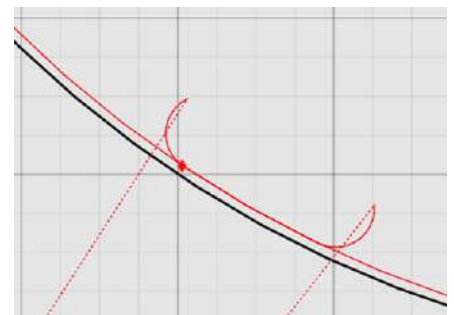
From here, you may click and drag the part to a specific location. You can also rotate parts by clicking and dragging one of the arrows in the corners. Moving a feature also adjusts automatically generated features, including lead in, lead out lines and rapid move paths.



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.

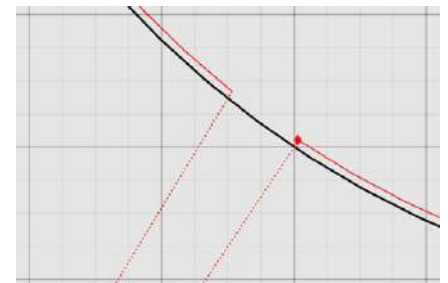
The initial locations of these lines are automatically generated by an internal algorithm using basic rules. FlashCut CAM enables you to override these initial settings for each individual feature.



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.

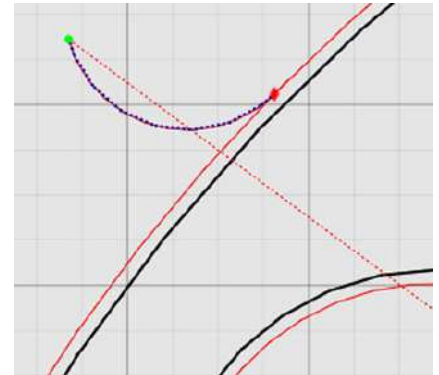
The bottom drawing shows a tab break, no lead in or lead out.

Manual editing

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

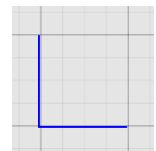
Click on the red diamond to make the line segment active. A blue dot appears at the end of the line. Hover over the blue dot until it turns green. This dot can now be used to drag the end of the lead in or lead out to the desired position.

The example at right shows a single lead in with its end point selected (cursor not shown for clarity).



Program zero

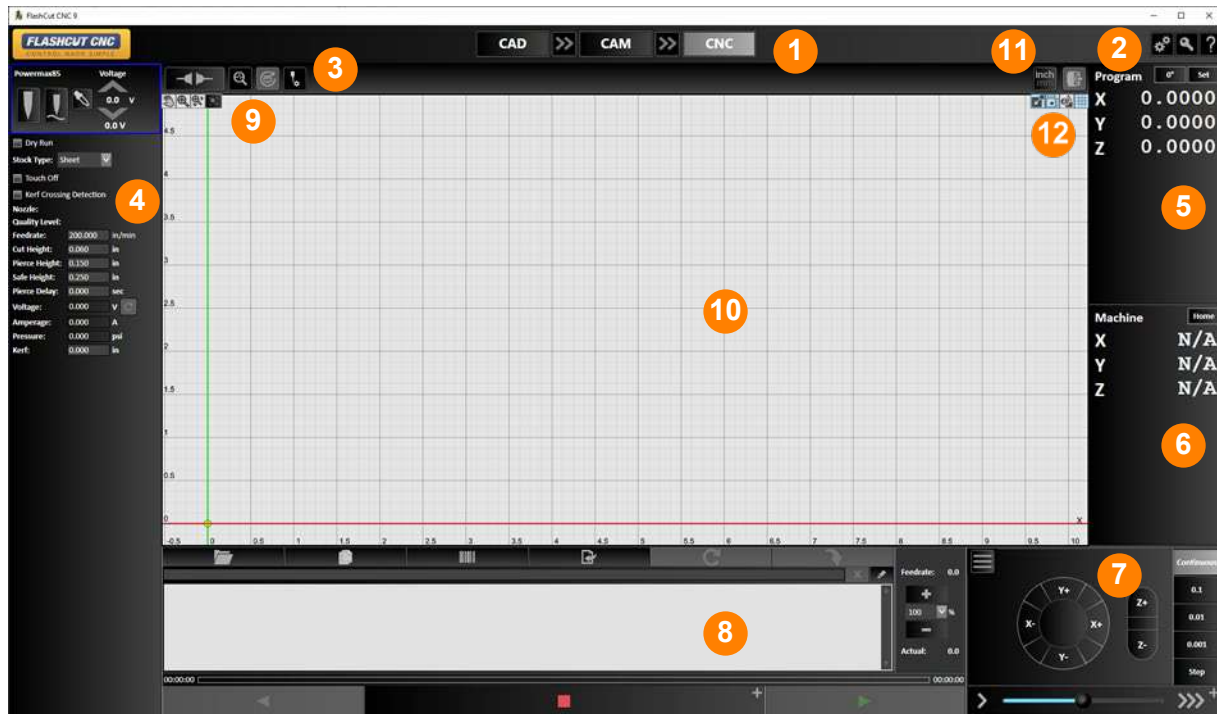
The blue set of axes in the drawing window represents the origin of the program coordinates. You may move these axes around by clicking and dragging the blue lines, or by entering coordinates manually in the parameter window.



FlashCut CNC

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

The main screen is shown below. An explanation of each area of the screen follows.



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

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

System Status, Connect, Reset Motor Drivers and Tool Rack Positions

System Status

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

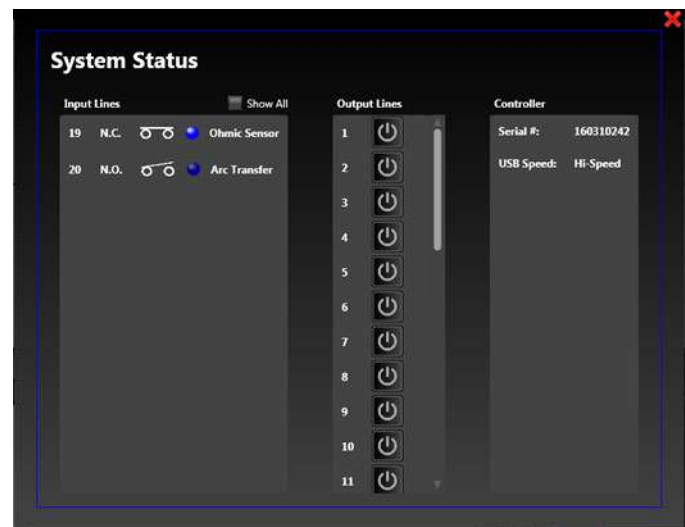


Input Lines

If a normally closed (NC) input line is **normal**, it will be in the **closed** state and the blue LED will **not** be illuminated. If the machine is connected, these icons are live when a switch changes its state.

If an NC switch is **tripped** then it will be in the open state and the blue LED will be illuminated.

Normally, only input lines that have been defined are displayed. However, if the **show all** checkbox is selected, the status of all input lines will be displayed.



Output Lines

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

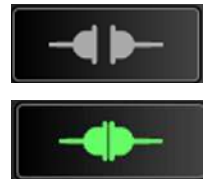
Controller

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

Connect

Clicking this icon connects or disconnects the signal generator. When connected, the icon is illuminated and the two halves are in contact.

Ensure that the signal generator (CNC controller) is securely connected via USB cable to your PC, and that the USB driver is installed (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.

When properly connected, the **Connect** icons will join and turn blue. Press **Disconnect** prior to unplugging the signal generator.

When the signal generator is connected, all moves are performed by the machine tool.

Before the unit connects, a safety reminder screen appears. It is imperative that you and anyone else near the machine understand, agree with and adhere to all of the safety guidelines. If the safety guidelines are not accepted, the software will not connect.

Reset Motor Drivers

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



Tool Rack Positions

Click to view and change tool rack positions



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



Fab head settings

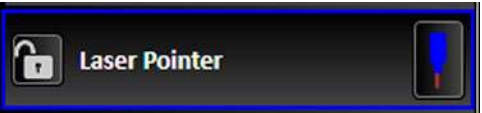
Each Fab Head that has been configured and saved will be displayed here. See [Configuring FlashCut](#) for more information.

If multiple Fab Heads are configured, the parameters for a specific Fab Head can be viewed and edited by selecting the appropriate tab.



Laser pointer control

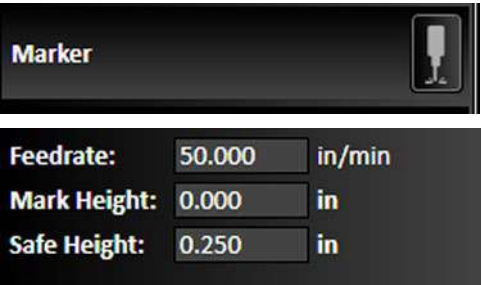
If a laser pointer is configured, it can be turned on and off by clicking the icon. The pointer can also be locked in the on position.



Marker control

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

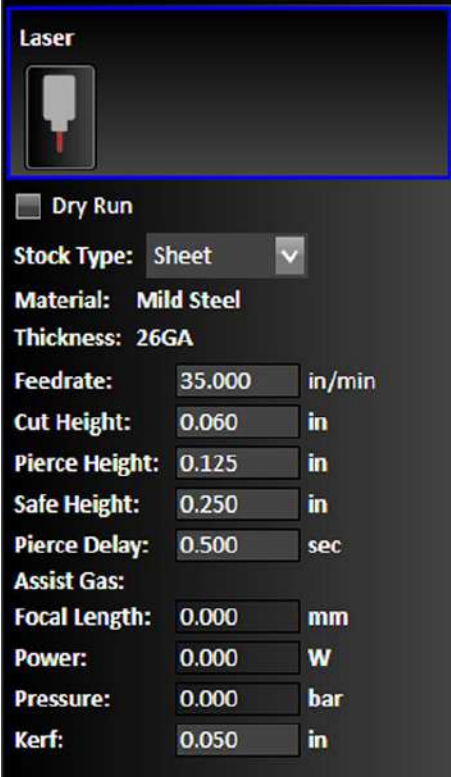
- **Feedrate** – in system units/min.
- **Mark Height** – Specifies the distance between the marker tip and the stock material throughout a marking operation.
- **Safe Height** – Specifies the height to which the Fab Head is set during rapid moves.



Laser cutter settings

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

- **Laser On/Off** – Click to turn the laser on or off. Before turning on the laser, FlashCut CNC will display a warning dialog asking if it is safe to proceed. Controls for other options (e.g., **Marking**) will appear here if configured.
- **Dry Run** – When selected, the machine will execute the operations, but the laser will not activate, and cuts will not be made.
- **Stock Type, Material** and **Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.
- **Cut Height** – Specifies the distance between the torch tip and the stock material throughout a cutting move.
- **Pierce Height** – Specifies the distance between the torch tip and the stock material at the time when the laser is activated.
- **Safe Height** – Specifies the height to which the laser is set during rapid moves.
- **Assist Gas** –
- **Focal Length** –
- **Power** –
- **Pressure** – Pressure (psi) of the Fab Head's assist gas jet.
- **Kerf** – Specifies the width of the kerf used by the cutter.



Laser

☐ Dry Run

Stock Type: Sheet ▼

Material: Mild Steel

Thickness: 26GA

Feedrate: 35.000 in/min

Cut Height: 0.060 in

Pierce Height: 0.125 in

Safe Height: 0.250 in

Pierce Delay: 0.500 sec

Assist Gas:

Focal Length: 0.000 mm

Power: 0.000 W

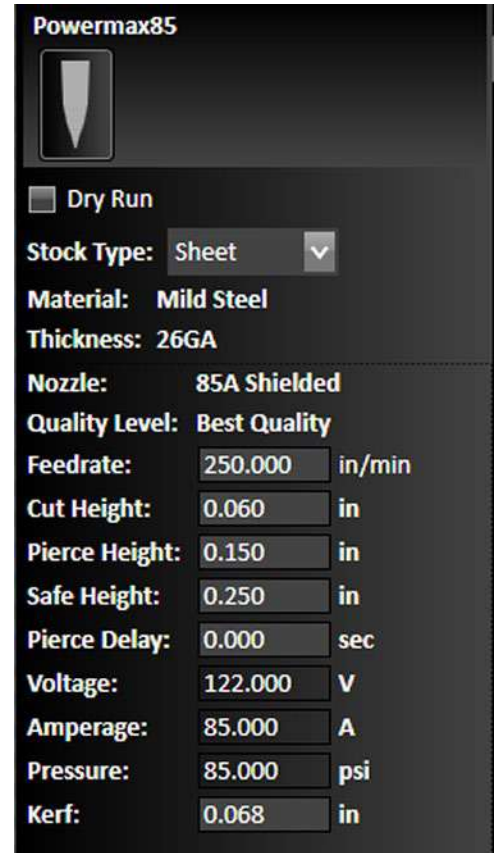
Pressure: 0.000 bar

Kerf: 0.050 in

Plasma torch settings

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

- **Torch On/Off** – Click to turn the torch on or off.
Before turning on the torch, FlashCut CNC will display a warning dialog asking if it is safe to proceed.
- **Dry Run** – When selected, the machine will execute the operations, but the torch will not be lit, and cuts will not be made.
- **Stock Type, Material and Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.
- **Nozzle** – Displays information about the plasma torch nozzle.
- **Quality Level** – Specifies whether the cut will be optimized for speed or quality.
- **Feedrate** – in system units/min.
- **Cut Height** – Specifies the distance between the torch tip and the stock material throughout a cutting move.
- **Pierce Height** – Specifies the distance between the torch tip and the stock material at the time when the torch is commanded to fire.
- **Safe Height** – Specifies the height to which the torch is set during rapid moves.
- **Pierce Delay** – Specifies the delay between the command to fire the torch and the start of the next move.
- **Voltage** – Specifies the operating voltage of the torch.
- **Amperage** – Specifies the operating amperage of the torch.
- **Pressure** – Specifies the air pressure of the torch in psi.
- **Kerf** – Specifies the width of the kerf used by the torch.



Powermax85

☐ Dry Run

Stock Type: Sheet ▾

Material: Mild Steel

Thickness: 26GA

Nozzle: 85A Shielded

Quality Level: Best Quality

Feedrate: 250.000 in/min

Cut Height: 0.060 in

Pierce Height: 0.150 in

Safe Height: 0.250 in

Pierce Delay: 0.000 sec

Voltage: 122.000 V

Amperage: 85.000 A

Pressure: 85.000 psi

Kerf: 0.068 in

Oxyfuel settings

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

- **Control Buttons** – For cutting and water
- **Preheat Delay** – Specifies the time to preheat the material before cutting.
- **Preheat Countdown** –
- **Preheat Time Elapsed** –
- **Stock Type, Material** and **Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.
- **Pierce Delay** – Specifies the time to delay before completing a pierce operation.
- **Purge Delay** – Specifies the time to delay before completing a purge operation
- **Use Water** – Toggles this feature, if enabled.
- **Kerf** – Specifies the width of the kerf used by the cutter.
- **Creep Rate** –
- **Creep Time** –

Oxyfuel

Icons: Torch, Flame

Preheat Delay: 0.0 sec

Preheat Countdown: 0.0 sec

Preheat Time Elapsed: 0.0 sec

Buttons: Release, Extend, Set and Release

☐ Dry Run

Stock Type: Sheet

Material: Mild Steel

Thickness: 26GA

Feedrate: 30.000 in/min

Pierce Delay: 0.000 sec

☒ Use Water

Kerf: 0.040 in

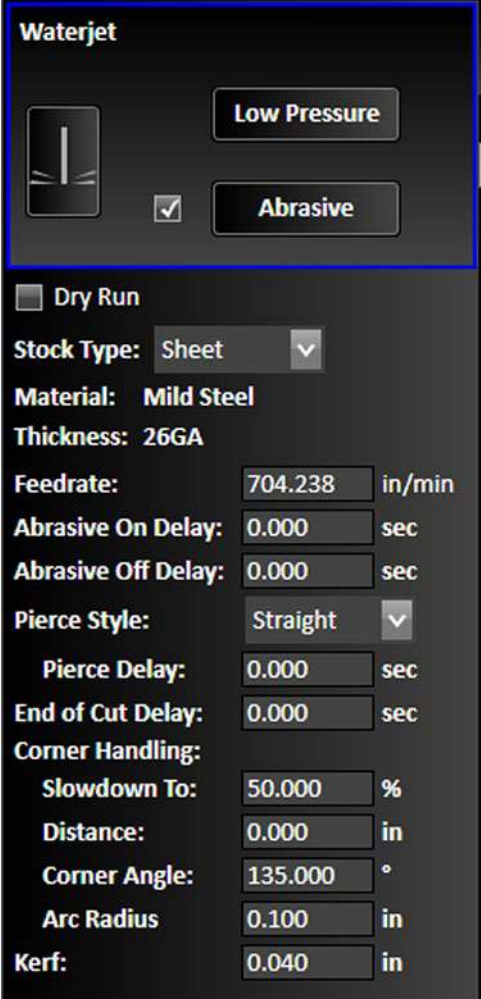
Creep Rate: 100 %

Creep Time: 0.000 sec

Waterjet settings

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

- **Control Buttons** – Click to turn the jet on or off. If configured, buttons to select **Low Pressure** and **Abrasive** will also be visible.
- **Dry Run** – When selected, the machine will execute the operations, but the jet will not be activated, and cuts will not be made.
- **Stock Type, Material and Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.
- **Feedrate** – in system units/min.
- **Abrasive On Delay** – Specifies the time in seconds before the flow of abrasive starts after the cutter is activated.
- **Abrasive Off Delay** – Specifies the time before the flow of abrasive ends when the jet is deactivated.
- **Pierce Style** – Straight is shown. Additional configuration options are visible for specific types (e.g., **Radius** for **Circular**)
- **Pierce Delay** – Specifies the time to delay before completing a pierce operation.
- **End of Cut Delay** – Specifies the delay at the end of the cut. The purpose of the delay is to make sure the cut is complete.
- **Corner Handling** – Settings to enable cleaner corner cuts: **Slowdown** percentage, **Corner Angle** and **Arc Radius**.
- **Kerf** – Specifies the width of the kerf used by the jet.

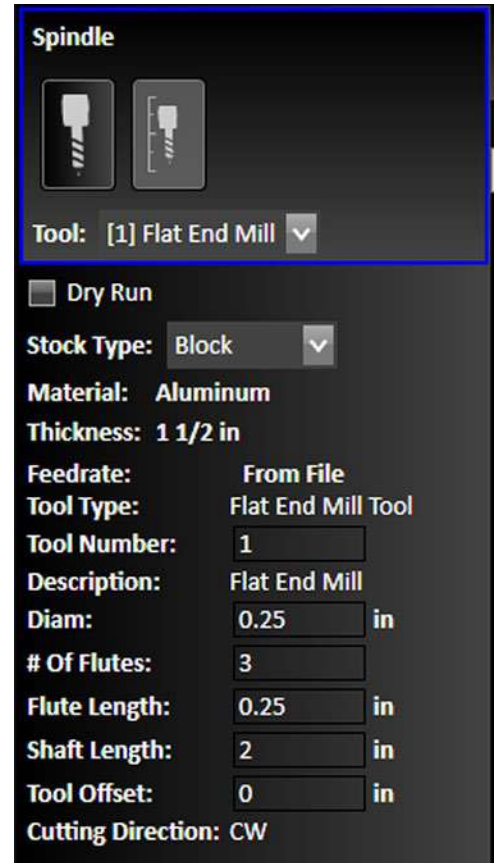


The screenshot shows the 'Waterjet' settings panel. At the top, there is a 'Waterjet' title and a 'Low Pressure' button. Below this is a 'Dry Run' checkbox, which is currently unchecked. To the right of the checkbox is a 'Sheet' dropdown menu. Below the dropdown is a 'Material' field set to 'Mild Steel' and a 'Thickness' field set to '26GA'. The 'Feedrate' is set to '704.238 in/min'. Below this are 'Abrasive On Delay' and 'Abrasive Off Delay' fields, both set to '0.000 sec'. The 'Pierce Style' is set to 'Straight' with a dropdown arrow. Below this are 'Pierce Delay' and 'End of Cut Delay' fields, both set to '0.000 sec'. The 'Corner Handling' section includes 'Slowdown To' set to '50.000 %', 'Distance' set to '0.000 in', 'Corner Angle' set to '135.000 °', and 'Arc Radius' set to '0.100 in'. The 'Kerf' is set to '0.040 in'.

Spindle settings

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

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



Spindle

Tool: [1] Flat End Mill ▼

☐ Dry Run

Stock Type: Block ▼

Material: Aluminum

Thickness: 1 1/2 in

Feedrate: From File

Tool Type: Flat End Mill Tool

Tool Number: 1

Description: Flat End Mill

Diam: 0.25 in

Of Flutes: 3

Flute Length: 0.25 in

Shaft Length: 2 in

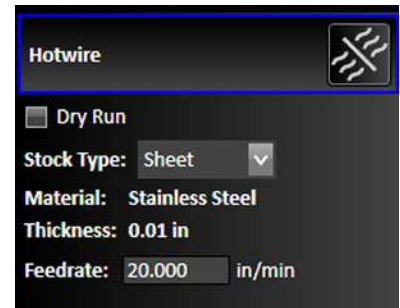
Tool Offset: 0 in

Cutting Direction: CW

Hotwire Settings

The HotWire Fab Head can only be used directly in FlashCut CNC. See [Configuring FlashCut](#).

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



Hotwire 

☐ Dry Run

Stock Type: Sheet ▼

Material: Stainless Steel

Thickness: 0.01 in

Feedrate: 20.000 in/min

Tangential Knife Settings

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

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

Knife



Tool: [2] Straight Knife ▼

☐ Dry Run

Stock Type: Sheet ▼

Material: Other

Thickness: 1/16 in

Feedrate: 100.000 in/min

Tool Type: Straight Knife

Tool Number: 2

Description: Straight Knife

Blade Angle: 28.080 °

Overall Length: 3.000 in

Cutting Length: 2.000 in

Width: 1.000 in

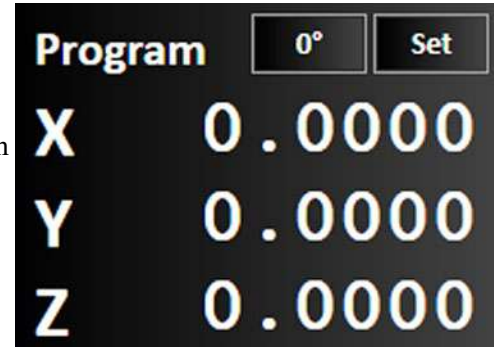
Program coordinates panel

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°)** sets the rotation of the drawing window
- **Set** sets the zero point



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 and point control panel**) and then selecting an item from the dropdown menu. Options include:
 - Lower Left and Lower Right
 - Lower Left and Upper Left
 - Lower Right and Upper Right
 - Upper Left and Upper Right
- **Enter Rotation Angle** allows manual entry of a numerical value in thousands of a degrees. The angle may be positive or negative.
- **Clear Rotation** resets the angle of rotation to zero degrees.

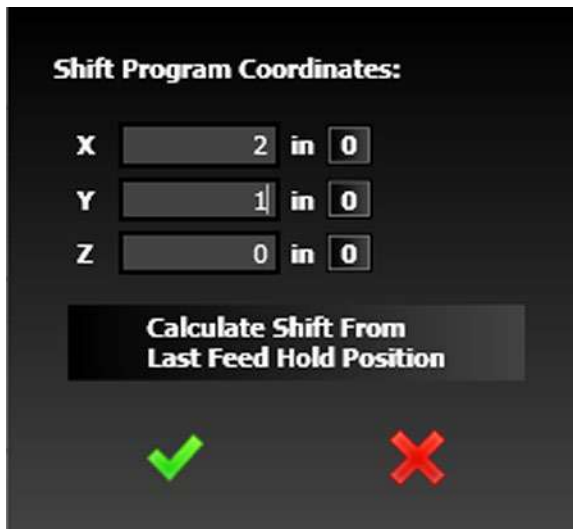
An example with step-by-step instructions appears below:

The **Set** dropdown menu sets program zero points along each axis, as selected from the dropdown menu.

- **Zero X** will zero the X axis.
- **Zero Y** will zero the Y axis
- **Zero X and Y** will zero both X and Y axes.
- **Zero Z** will zero the Z axis.
- **Zero All** will zero each program axis.
- **New** sets the position of the tool relative to program zero.
- **Shift** sets the position of the tool relative to the last feed hold position
- **Level Rotating Stock**

- **Select Offset**
- **Define Offset**

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



Shift Program Coordinates:

X	2	in	0
Y	1	in	0
Z	0	in	0

Calculate Shift From Last Feed Hold Position

✓ ✗

Coordinate rotation example

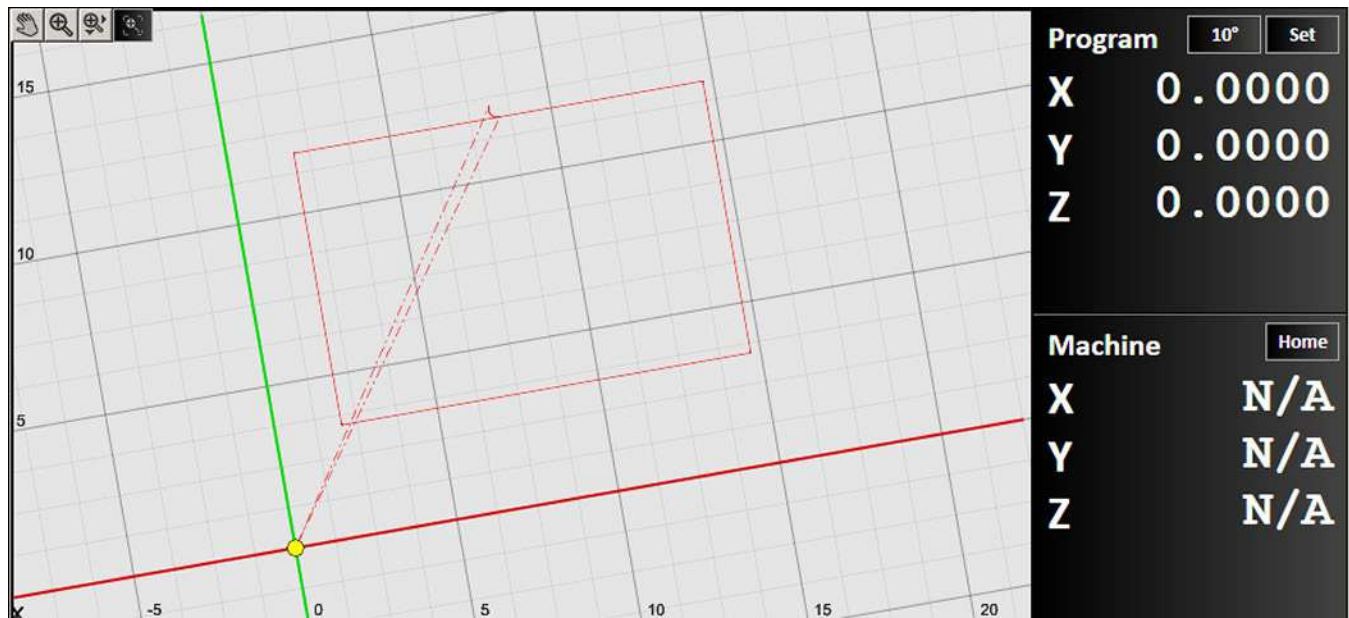
Coordinates rotation can be used to accommodate a workpiece that is too heavy or inconvenient to move on the machine. In effect, the program axes can be adjusted to conform to the dimensions and orientation of the workpiece.

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.

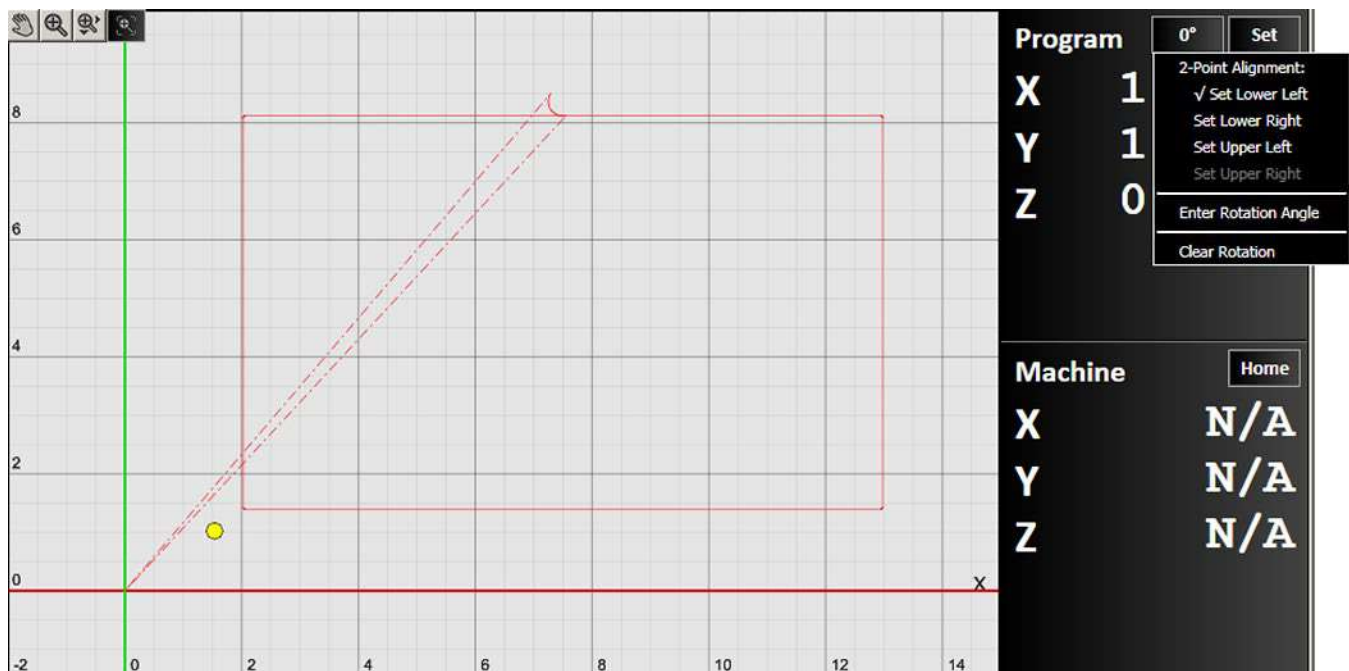
The angle of rotation will be displayed in the Program coordinates panel, and the drawing workspace will be rotated.



Use 2-point alignment

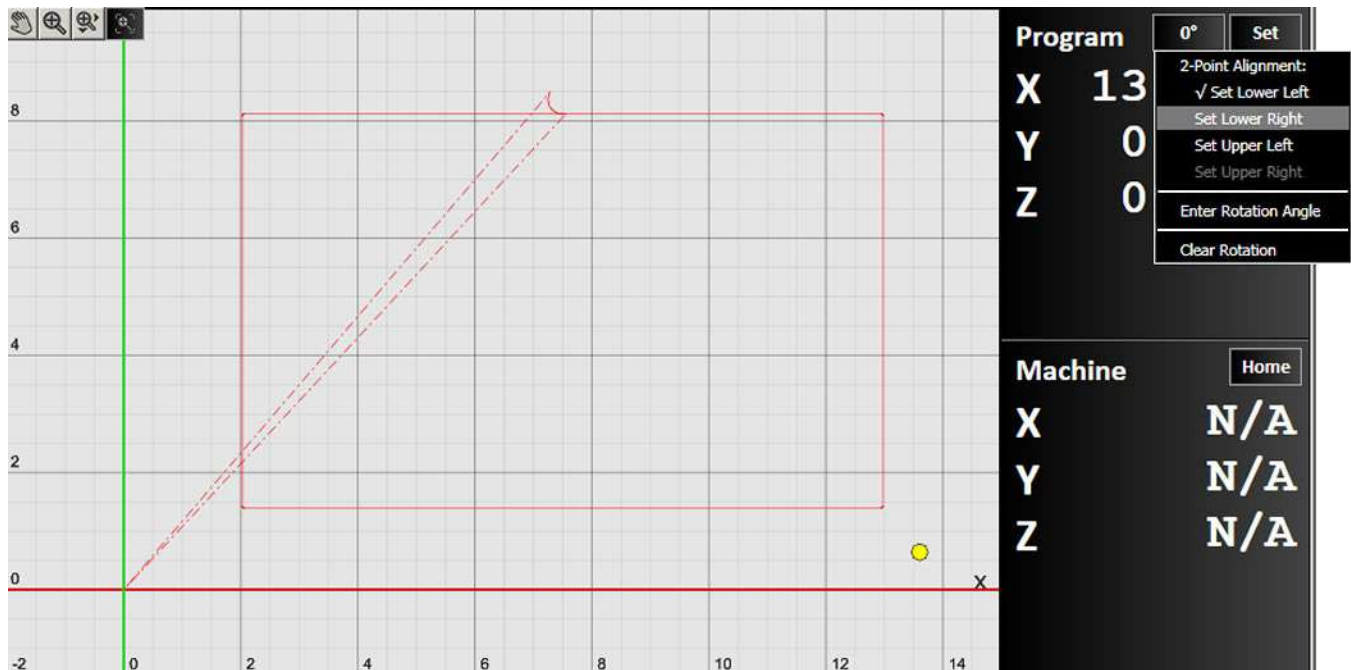
When using 2-point alignment, the position of the tool is used to define two points on the workpiece that define the angle of rotation. The angle of rotation is calculated by comparing the relative positions of two points selected.

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.

Machine		Home
X	0.0000	
Y	0.0000	
Z	0.0000	

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.

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 and point control panel

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



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.

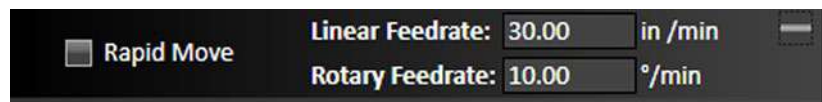
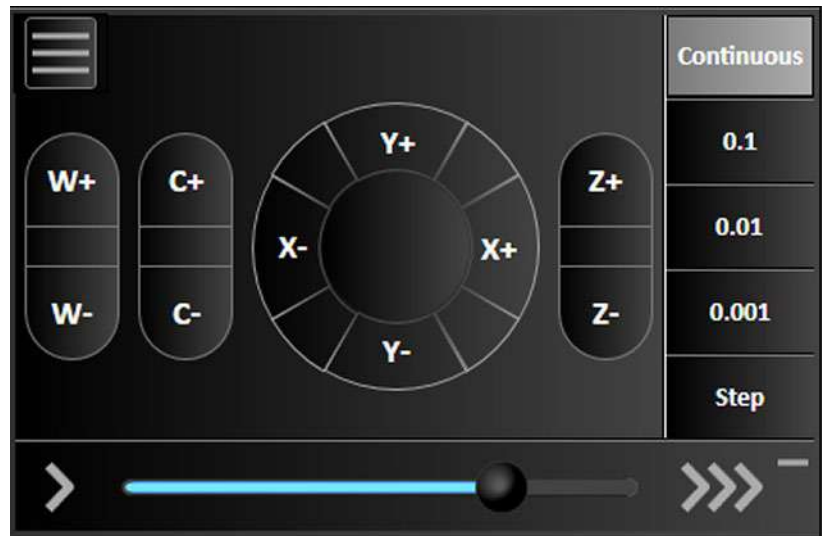
Axis jog buttons

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

Diagonal jog buttons

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

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



Jog mode

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

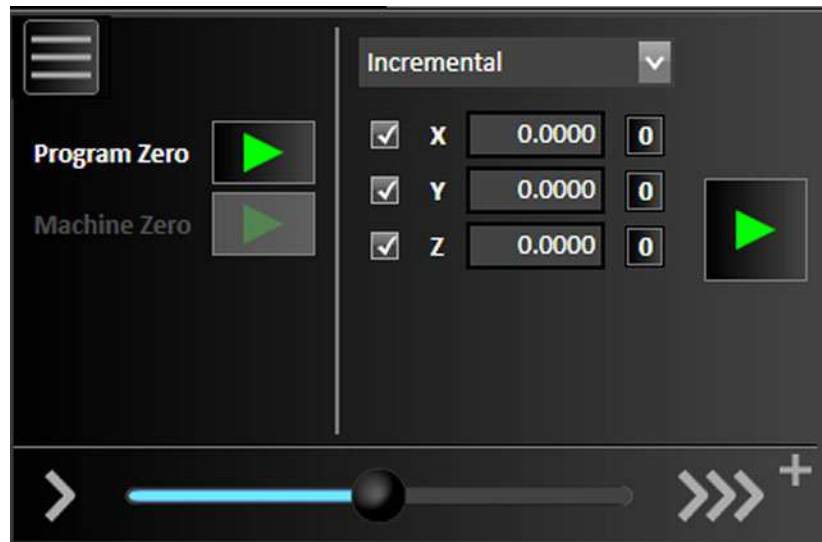
Jog rate

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.

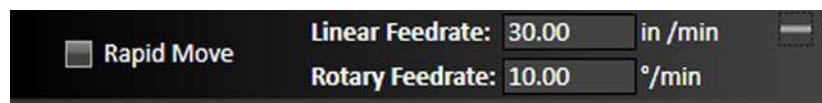
Move to point

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 **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

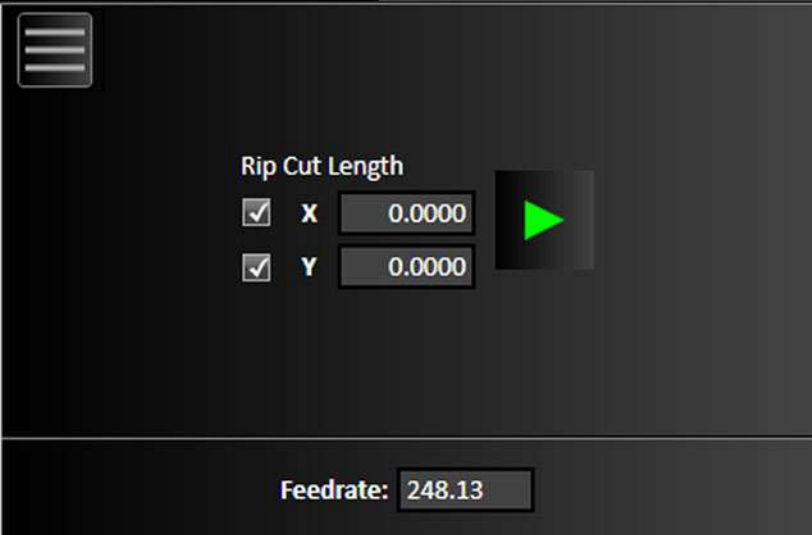
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

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.



The screenshot shows a dark-themed control panel for a 'Rip Cut'. In the top-left corner, there is a menu icon consisting of three horizontal lines. The main section is titled 'Rip Cut Length' in red text. Below this title, there are two rows of controls. The first row has a checked checkbox, the letter 'X', and a numeric input field containing '0.0000'. The second row has a checked checkbox, the letter 'Y', and a numeric input field containing '0.0000'. To the right of these input fields is a large green right-pointing triangle, which serves as the execution button. At the bottom of the panel, there is a label 'Feedrate:' followed by a numeric input field containing '248.13'.

Rip Cut Length	
☒ X	0.0000
☒ Y	0.0000

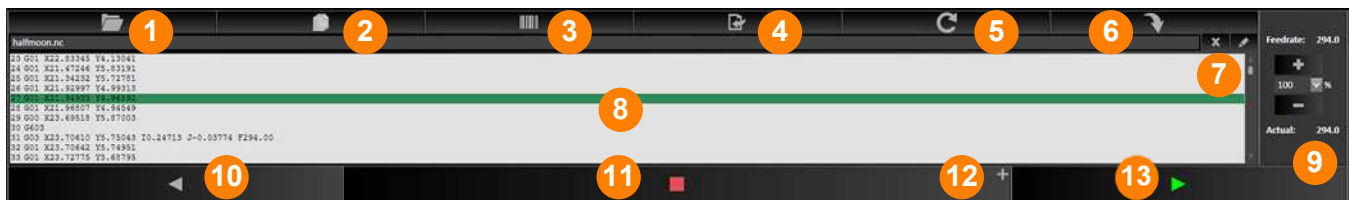
Feedrate: 248.13

G-Code window

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

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

This section describes these G-Code window features:



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

Open G-Code file

When a CAD or CAM design is sent to be manufactured in FlashCut CNC, it appears in the **G-Code workspace**. However, to open an existing program manually, click the **Open G-Code file** button to browse to a G-Code file and open it in the **G-Code workspace**. A visual preview of the code will appear in the workspace. The name of the G-Code file is displayed in the **G-Code workspace**.

This workspace displays the currently loaded G-Code file. While the program is running the current line of code is highlighted in real time, this way the user can track their progress throughout the operation.

File name

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

Load all G-Code files in a folder

Prompts the user to select a folder. All G-Code files in the folder will be loaded into FlashCut CNC.

Barcode

Allows the user to input a barcode representing a list of files to be cut.

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 workspace

This workspace displays the currently loaded G-Code file. While the program is running the current line of code is highlighted in real time, this way the user can track their progress throughout the operation.

Note Double-clicking in the G-Code workspace now brings up the G-Code editor.

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 troubleshooting a program in simulation mode or dry run mode. Not typically used while actually 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 mode

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



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



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

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**.



Pan and zoom tools



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

Pan



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

Zoom Extent



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

Zoom



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

Zoom to Fit

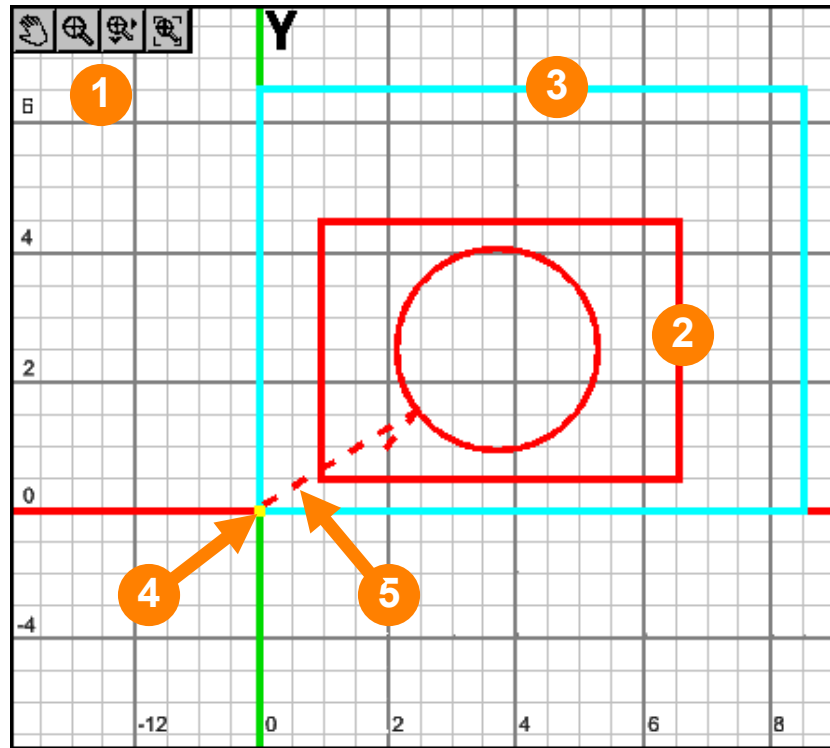


Click this button to adjust zoom automatically to best fit the drawing.

Drawing workspace

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

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



1 Pan and zoom tools

4 The current tool position is shown by a yellow dot.

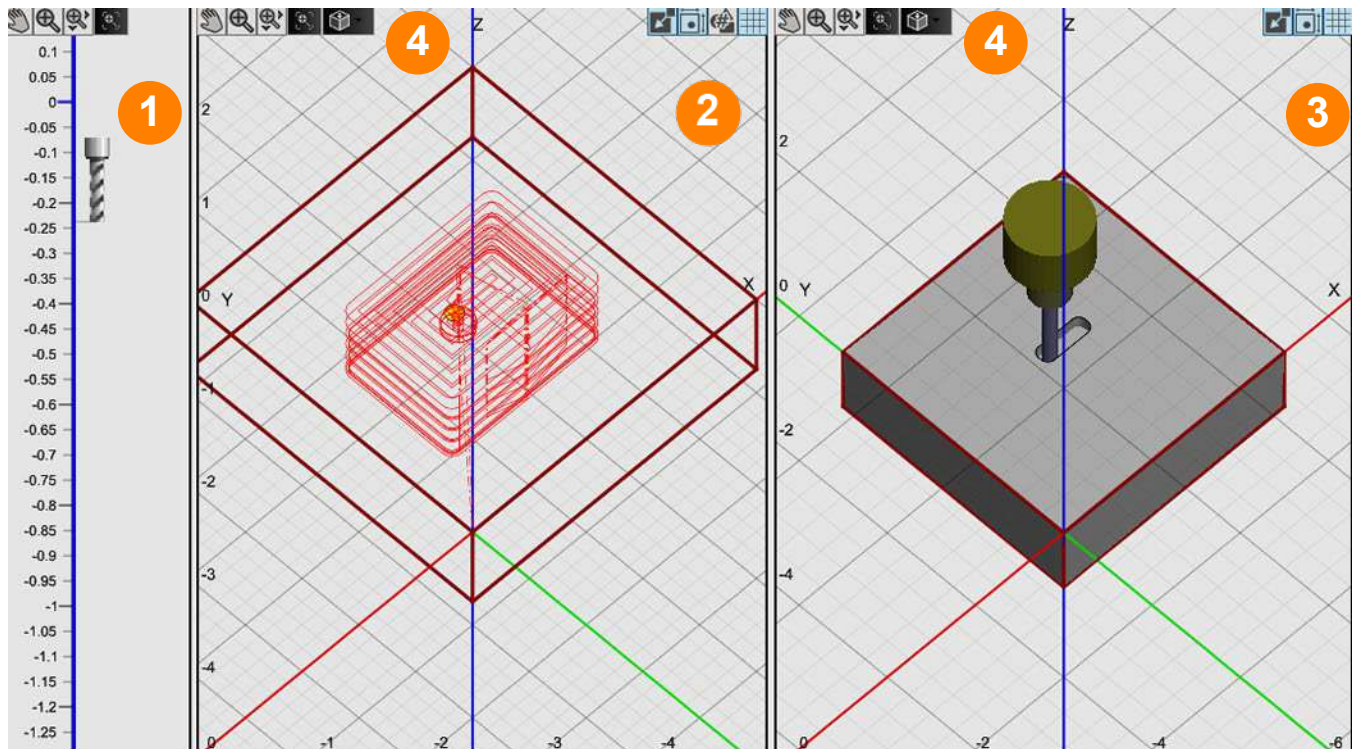
2 Solid red lines show the toolpath

5 Dotted red lines represent rapid moves

3 The maximum dimensions which a tool may not exceed are represented by a thick blue box. The box will appear after the machine has been homed. See **Machine Envelope**.

To avoid improper machining, ensure that the entirety of the toolpath remains within this machine tool envelope. If the feature dimensions exceed this size, return to FlashCut CAD or FlashCut CAM in order to make revisions.

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.

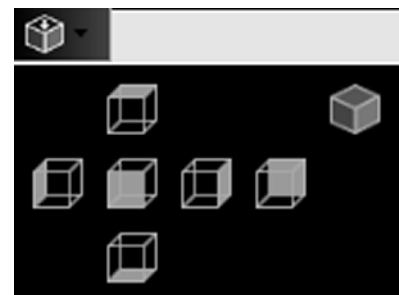
3 Simulation view

Displays a simplified solid model of the workpiece and tool.

Orientation tool

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

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



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.



Toggle manual control

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

This icon is greyed out on machines without this capability.



Material

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



Machine Envelope

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



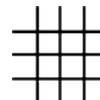
Part Numbers

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



Show/Hide Grid

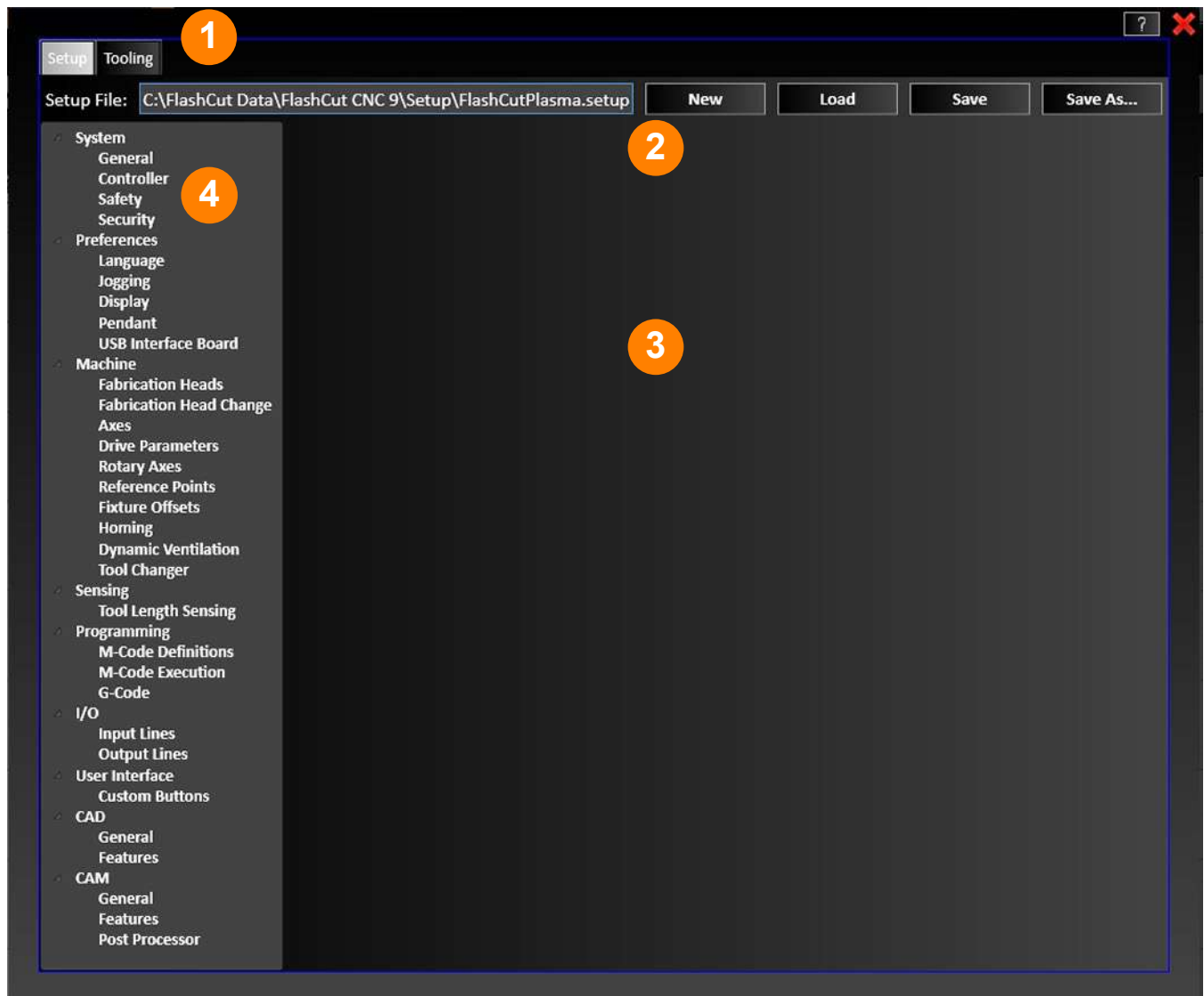
Toggles the **grid** in the active drawing workspace.





The **Configuration** icon is available in all three windows: CAD/CAM/CNC.

Click the icon to examine or modify FlashCut settings.



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

- | | |
|--|--|
| <p>1 Menu.selection.tab</p> <p>3 Configuration workspace</p> | <p>2 Menu.selection.tab</p> <p>4 Configuration workspace</p> |
|--|--|

Menu selection tab

Select the appropriate tab to configure different FlashCut settings.

Setup Tab

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

Tooling Tab

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

Setup file (*.setup extension)

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

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

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

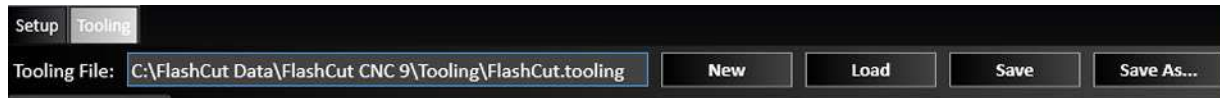
The **New** button allows you to duplicate a previously-created setup file. Select the file to be copied then click **OK**. Note that this is a temporary copy of the file that must be saved in order to solidify any changes; it will not affect the original file.

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

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

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

Tooling file (extension *.tooling)



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

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

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

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

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

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

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

Configuration workspace

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

Setup Configuration Menu

- 1** System
 - General
 - Controller
 - Safety
 - Security
- 2** Preferences
 - Language
 - Jogging
 - Display
 - Pendant
 - USB Interface Board
- 3** Machine
 - Fabrication Heads
 - Fabrication Head Change
 - Axes
 - Drive Parameters
 - Rotary Axes
 - Reference Points
 - Fixture Offsets
 - Homing
 - Dynamic Ventilation
 - Tool Changer
- 4** Sensing
 - Tool Length Sensing
- 5** Programming
 - M-Code Definitions
 - M-Code Execution
 - G-Code
- 6** I/O
 - Input Lines
 - Output Lines
- 7** User Interface
 - Custom Buttons
- 8** CAD
 - General
 - Features
- 9** CAM
 - General
 - Features
 - Post Processor

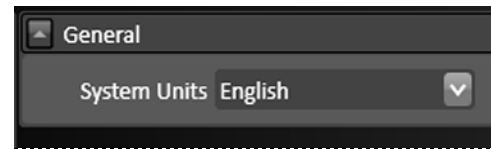
- System
 - 1** General
 - Controller
 - Safety
 - Security
- Preferences
 - Language
 - 2** Jogging
 - Display
 - Pendant
 - USB Interface Board
- Machine
 - Fabrication Heads
 - 3** Fabrication Head Change
 - Axes
 - Drive Parameters
 - Rotary Axes
 - Reference Points
 - Fixture Offsets
 - Homing
 - Dynamic Ventilation
 - Tool Changer
- Sensing
 - Tool Length Sensing **4**
- Programming
 - 5** M-Code Definitions
 - M-Code Execution
 - G-Code
- I/O
 - 6** Input Lines
 - Output Lines
- User Interface
 - Custom Buttons **7**
- CAD
 - 8** General
 - Features
- CAM
 - 9** General
 - Features
 - Post Processor

General

System Units

Use the **System Units** drop-down to choose which units of measurement to use. You may choose:

- English
- Metric



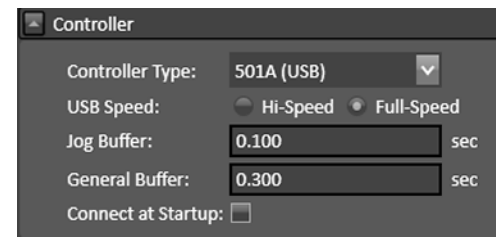
Controller

- Controller
- Motor Signals
- Signal State Preferences

Controller

The **Controller** settings section presents the following controls:

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



Controller Type

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

- 501A (USB connection)

USB Speed

USB Speed allows you to specify the speed of USB communications with the signal generator. Setting the USB speed to **Full Speed** helps to prevent communication problems.

Jog Buffer (sec)

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 (sec)

General Buffer (sec) sets the buffer size on the signal generator for all operations except jogging in continuous mode.

Connect at Startup

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

Motor Signals

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

The screenshot shows the 'Motor Signals' configuration window. It contains the following settings:

Parameter	Value	Unit
Controller Model:	FlashCut 2.5A Compact Micro Stepper (5501-x-025-M)	
Drive Type:	Stepper	
Step Pulse Polarity:	High	
Step Pulse Activation:	Normal	
Step Pulse Width:	2	µsec
Min Time Between Steps:	2	µsec
Direction-Step Setup:	2	µsec
Min Step-Direction Lag:	2	µsec
Enable Signal Polarity:	High	
Dynamic Enable Line	☐	

Driver 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).

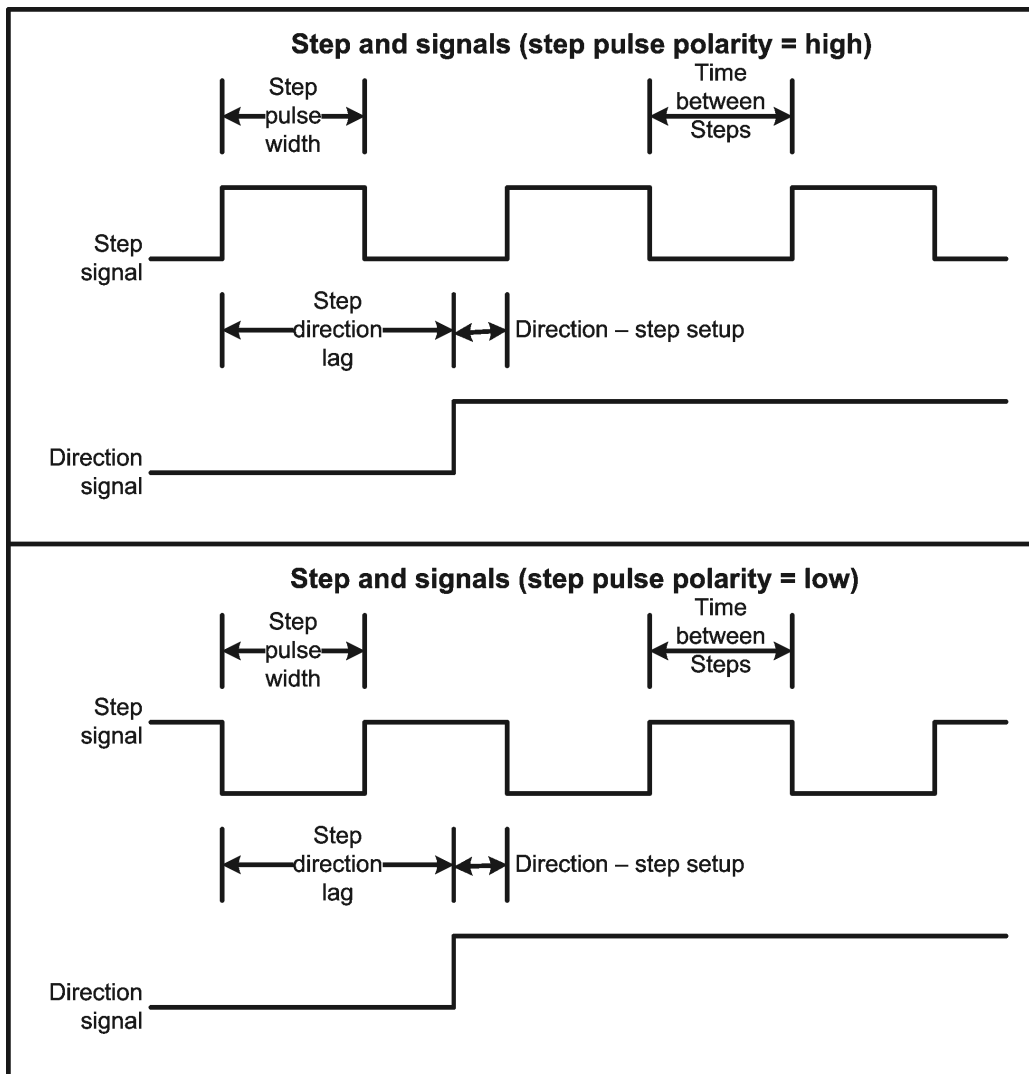
If you are using non-FlashCut drivers, choose **Other**. This enables all fields, allowing you to enter the proper settings based on your motor driver requirements. Note that some signal fields do not apply to all signal generator models, and may be disabled based on your controller settings (signal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

Driver 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 Activation

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

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

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

Optimize – FlashCut automatically selects the best activation mode.

Step Pulse Width (µsec)

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

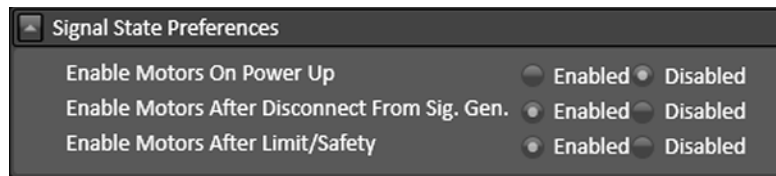
Sets the polarity of the enable signal. Choose **High** if the motor driver is enabled by a high signal, or **Low** if the motor driver is enabled by a low signal.

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~~
- ~~Enable Motors After Disconnect From Sig. Gen.~~
- ~~Enable Motors After Limit/Safety~~



Enable Motors On Power Up

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

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

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

Automatic Safety Response to Input Lines Tripped

During Motion

Input Line Types: Saftey, Limit

System Response

☐ Emergency Shut Off All Fabrication Heads

Set Output Lines:

+

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

Safety Interlocks (Invalid Output Line Combinations)

Line #	Set To	Line #	Set To
1	On	2	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On
None	On	None	On

Security

- ~~Use Administrator Password~~
- ~~Password and Password Confirmation~~

Security

Use Administrator Password ☐

Password: (a-z, 0-9)

Password Confirmation:

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**
- **Jogging**
- **Display**
- **Pendant**
- **USB Interface Board**

Language

Preferred Language



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

- English (United States)
- Spanish (Spain)
- Italian (Italy)
- French (France)
- Portuguese (Portugal)

Jogging

The **Jogging** configuration option displays the following controls:

- **Jog and Point Move Rates**
 - **Jog Distances**
 - **General**
-

Jog and Point Move Rates

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

Slow

This column corresponds to the feedrate specified in the **Jog and point control panel**.

FlashCut executes all jog moves using the rates you enter in these fields.

Jog and Point Move Rates		
Axis	Slow	
X	30	in /min
Y	30	in /min
Z	10	in /min
W	10	in /min
V	10	in /min
C	10	°/min

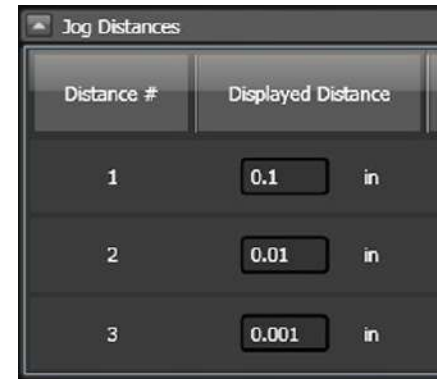
Jog Distances

Distance

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

Displayed Distance

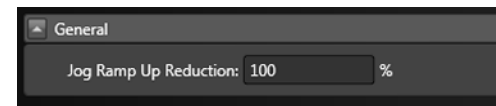
This is the value of the incremental move that is commanded each time the user clicks a directional jog button on the CNC panel.



General

Jog Ramp Up Reduction

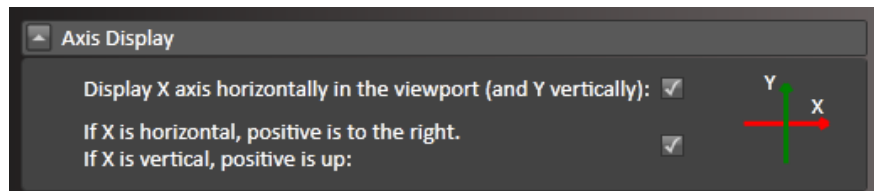
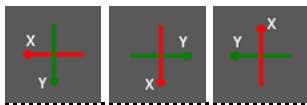
Set the percentage of the speed at which the rate increases during jog operations. The default is 100%.



Display

Axis Display

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



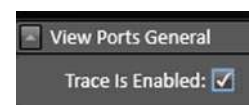
Current Position

Select the desired icon to mark the current position.



View Ports General

Select to enable Trace in viewports.



Pendant

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

General

Use Pendant

Enables control of the machine from the pendant.

Number of Jog Rates

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

Model

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

Pendant Buttons

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

General

☒ Use Pendant

Number of Jog Rates: 3 Model: 20-Key

Layer: Green Red

Layer	Button	Function
Green	1	Jog: Z+
Green	2	Jog: Y+
Green	3	Jog: X-
Green	4	Jog: X+
Green	5	Jog: Z-
Green	6	Jog: Y-
Green	7	Jog: A/W-
Green	8	Jog: A/W+
Green	9	Jog Rate: 3
Green	10	Jog Rate: 2
Green	11	Jog Rate: 1 (Slowest)
Green	12	Jog: Continuous

Pendant buttons:

G-Code Files to Load and Start

#	File
---	------

Use Standard Plasma Layout

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

Functions

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

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

Use USB Interface Board

Enables control of the machine from the board.

Number of Jog Rates

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

Functions

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

Each function has four attributes:

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

Layer	Wiring Row	Wiring Column	Function
Green	1	1	Jog: X-
Green	1	2	Jog: X+
Green	1	3	Jog: Y-
Green	1	4	Jog: Y+
Green	1	5	Jog Rate: 1 (Slowest)
Green	1	6	Jog Rate: 2
Green	1	7	Jog Rate: 3

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.

Machine

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

Fabrication Heads

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

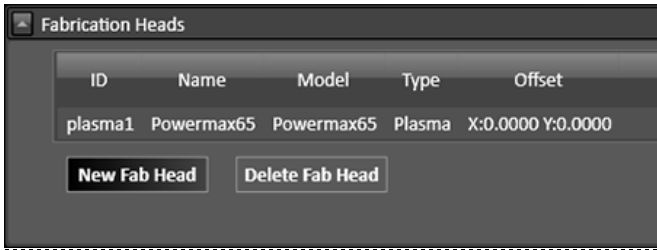
- Fabrication Heads
- Cut Charts

Fabrication Heads

Fabrication heads are the actual cutting heads on your machine.

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

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



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

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

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

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

Plasma

- **Type** and **SubType** – When Plasma is the selected type, a dropdown list of subtypes will be displayed.
- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Model** – Select from the drop down list.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Lift Axis Offset** – Enter offset for selected lift axis.
- **Communication** – Select RS-232/RS-485 and a Comm port if you have direct communication with the plasma for setting Current and Pressure on the fly.
- **Pause G-Code on Mode Change** – Select to enable.
- **Torch Control**
 - Torch On/Off Output Lines** – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.
 - Custom Macro** – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting **Edit** will bring up the code editor window.

Type: Plasma
SubType: Powermax

General

ID#: 1

Model: Powermax85

Name: Powermax85

Offset: X: 0 in
Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Communication: ☐ None ☐ RS232/RS422/RS485

Pause G-Code on Mode Change: ☐

Torch Control

Torch On: Output Lines: 1 Delay: 0 sec ☐ Custom Macro

Torch Off: Output Lines: 1 Delay: 0 sec ☐ Custom Macro

- **Cutting**

Start/End of Cut – When selected displays operations to be executed.

Selecting **Edit** will bring up the code editor window.

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

- **Torch Height Control**

Use Torch Height Control –

Select this value to enable torch height control, if your torch has it.

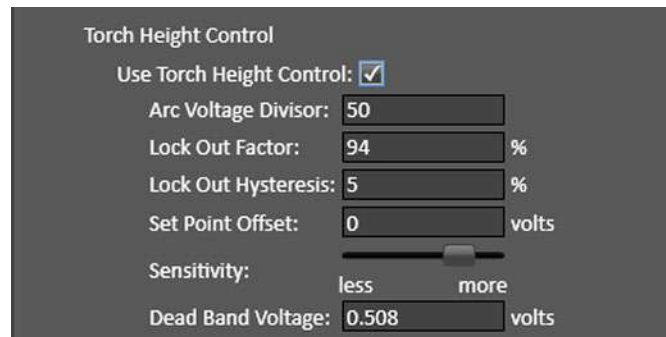
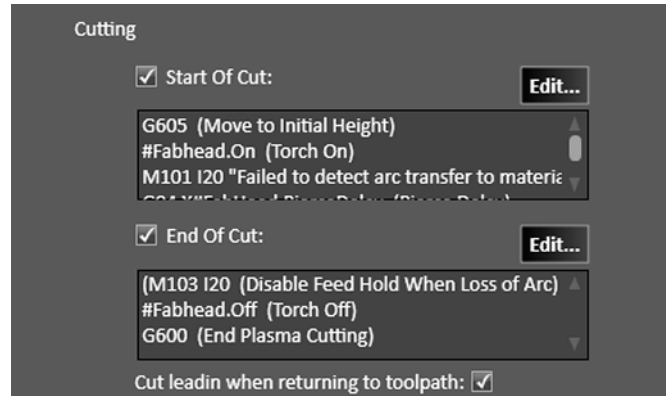
Arc Voltage Divisor – This is typically 50, and must match the exact voltage divider setting in your torch. Note that in order for the torch height control system to work properly, $(\text{maximum raw voltage}) / (\text{arc voltage divisor})$ must be smaller than 6V.

Lock Out Factor - Turns the THC off to prevent unwanted diving when the XY velocity of the torch is below this percentage of the programmed feedrate.

Lock Out Hysteresis - The THC will turn back on again when the XY velocity goes above the $((\text{Lock Out Factor}) + (\text{Lock Out Hysteresis})) * (\text{Programmed Feedrate})$.

Set Point Offset – Offsets the small residual voltage when torch is not in use.

Sensitivity - Determines how sensitive the THC acts towards voltage fluctuation. When the slider bar is adjusted, the new value appears in the **Dead Band Voltage** window below.



- **General Sensing**

Direction – Select direction.

Feedrate – Enter value in in/min.

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

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

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

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

The image shows two software control panels. The top panel, titled 'General Sensing', includes settings for Direction (radio buttons for Positive and Negative, with Negative selected), Feedrate (a numeric input set to 10 in/min), Sensing Method (a dropdown menu set to Monitor Input Line), and Primary Sensor settings (Monitor Input Line dropdown set to 19, Sensor Offset input set to 0 in). It also has a checkbox for 'Use Additional Sensor' which is checked, with its own Monitor Input Line dropdown set to 1 and Sensor Offset input set to 0 in. The bottom panel, titled 'Program Zero Sensing', has a checkbox for 'Use Program Zero Sensing' which is checked, and a 'Retract Distance' input set to 0.25 in.

- **Program Zero Sensing**

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

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

- **Additional Controls** – Check the box to enable a control. Specific values for these controls are set in the CNC window.

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

The image shows the 'Additional Controls' panel. It has a 'Safe Height' checkbox which is checked. Below it, the 'Specify As:' section has two radio buttons: 'Program Coordinate' (selected) and 'Incremental Distance'.

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

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

The image shows the 'Touch Off' panel with four checked checkboxes: 'Touch Off At Start (G605):', 'Zero Lift Axis Program Coord:', 'Rapid Move to Start Height:', and 'Enable Suppression Radius:'.

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

Rapid Move to Start Height – Select to enable rapid moves

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

Oxyfuel

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

Cut On/Off Output Lines –

When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

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

Support Automatic Ignition –

Enables this feature. M-Codes required.

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

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

Purge Delay – Enables this feature.

Type: Oxyfuel

General

ID#: 1

Name: Oxyfuel 1

Offset: X: 0 in, Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Allow Runtime Jog: ☐

Link To: None

Oxyfuel Control

Cut On: Output Lines: On

☒ Custom Macro Edit...

Cut Off: Output Lines: Off

☒ Custom Macro Edit...

Support Automatic Ignition: ☒

Ignition On: M

Ignition Off: M

Preheat Mode: Basic

Support Preheat: ☒

Preheat On: M

Preheat Off: M

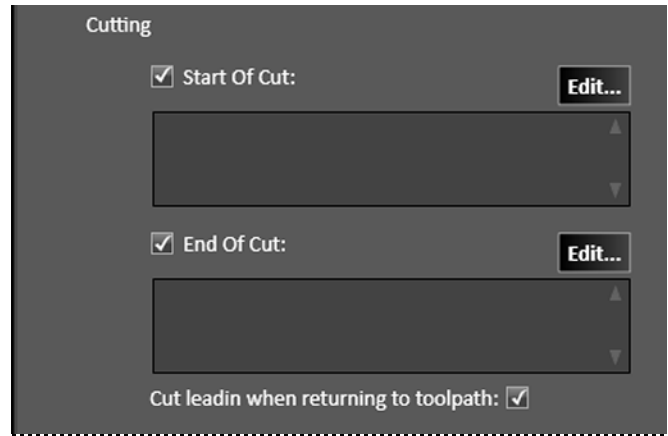
Support Use of Water: ☒

Water On: M

Water Off: M

Support Purge Delay: ☒

- **Cutting**
 - **Start/End of Cut**– When selected displays operations to be executed. Selecting **Edit** will bring up the code editor window.
 - **Cut lead in when returning to toolpath** – Check to allow recovery from a lost torch.



The screenshot shows a 'Cutting' dialog box with a dark gray background. It contains two main sections, each with a checked checkbox and an 'Edit...' button. The first section is 'Start Of Cut:' with a large text area below it. The second section is 'End Of Cut:' with a large text area below it. At the bottom of the dialog, there is a checkbox labeled 'Cut leadin when returning to toolpath:' which is also checked.

Cutting

☒ Start Of Cut: Edit...

☒ End Of Cut: Edit...

Cut leadin when returning to toolpath: ☒

Waterjet

When defining a new Waterjet head, select the **Type** from the drop down lists.

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

Waterjet On/Off Output Lines

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

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

Pump Pressure – Enter the pump pressure in PSI here.

Support Low Pressure Mode

– When checked, enter the proper M-Codes for Low Pressure On/Off. Note that the M-Code must be defined in the M-Code Definitions.

On/Off Delay After – Time in seconds

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

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

- **Cutting**

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

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

- **General Sensing**

Direction – Select direction.

Feedrate – Enter value in units/min.

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

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

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

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

- **Program Zero Sensing**

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

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

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

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

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

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

Rapid Move to Start Height – Select to enable rapid moves

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

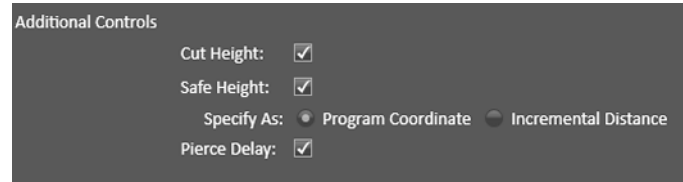
- **Additional Controls** – Check the box to enable a control.

Cut Height – Select to enable

Safe Height – This value can be specified as **Program**

Coordinate or **Incremental Distance**

Pierce Delay – Select to enable



Additional Controls

Cut Height: ☒

Safe Height: ☒

Specify As: ☒ Program Coordinate ☐ Incremental Distance

Pierce Delay: ☒

Laser

- **ID and Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.

Type: Laser

General

ID#: 1

Name: Laser 1

Offset: X: 0 in
Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Allow Runtime Jog: ☐

- **Link To** – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the drop down menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.
- **Laser Control**

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

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

Laser Control

Laser On: Output Lines: 1 On ☒ Custom Macro Edit...

Laser Off: Output Lines: 2 Off ☒ Custom Macro Edit...

- **Cutting**
- **Start/End of Cut** – When selected displays operations to be executed. Selecting **Edit** will bring up the code editor window.
- **Cut lead in when returning to toolpath** – Check to allow recovery from a lost torch.

Cutting

Enable: ☒

☒ Start Of Cut: Edit...

☒ End Of Cut: Edit...

Cut leadin when returning to toolpath: ☒

- **Marking**

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

Marking

Enable: ☒

☒ Start of Mark: Edit...

☒ End of Mark: Edit...

- **Positioning Laser**

Use Positioning Mode – Enables this feature.

Turn On/Off – M-Codes required.

Start/End of Positioning – M-Codes required.

Positioning Laser

Use Positioning Mode: ☒

Turn On: M

Turn Off: M

Start Of Positioning: M

End Of Positioning: M

- **Height Control**

Use Height Control – Select to enable.

Voltage Scale Factor – Number

Lock Out Factor – Percentage

Lock Out Hysteresis – Percentage

Idle Voltage Offset – Select to enable.

Sensitivity – Slider controls sensitivity.

Dead Band Voltage – Value calculated automatically from Sensitivity.

Height Control Calibration – Enter M-Code.

Calibration Output Line – Enter M-Code.

Height Control

Use Height Control: ☒

Voltage Scale Factor:

Lock Out Factor: %

Lock Out Hysteresis: %

Idle Voltage Offset: volts

Sensitivity:

Dead Band Voltage: volts

Height Control Calibration: M

Calibration Output Line: M

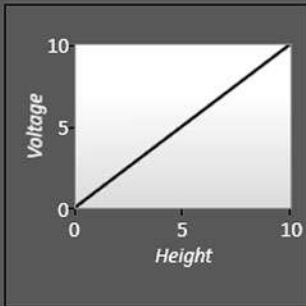
Pre Collision Offset: in

Measuring Range

Measuring Range Limit Units:

Minimum Height mm at volts

Maximum Height mm at volts



Pre Collision Offset – Enter distance.

- **Measuring Range**

Measuring Range Limit Units – Select English or metric.

Minimum/Maximum Height – Enter distance and voltage.

- **Laser Power**

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

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

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

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

Minimum and Maximum Laser Power – Enter watts, voltage.

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

Laser Power

Control Laser Power: ☒

Port Type: Analog (0-10V) ▼

Port Number: 1 ▼

Cut Power

Minimum Laser Cut Power: 0 watts at Voltage 0 volts

Maximum Laser Cut Power: 1500 watts at Voltage 10 volts

Mark Power

Minimum Laser Mark Power: 0 watts at Voltage 0 volts

Maximum Laser Mark Power: 300 watts at Voltage 10 volts

Use Dynamic Adjustment: ☒

Laser Power: 0 % at Feedrate 0 %

Laser Power: 100 % at Feedrate 100 %

% of Power

% of Feedrate

- **Focusing Lens**

Use Focusing Lens – Select to enable.

Focusing Lens Calibration – Enter M-Code.

Execute Focal Length – Enter M-Code.

Focusing Lens

Use Focusing Lens: ☒

Position Units: Metric ▼

Focusing Lens Calibration: M

Execute Focal Length: M

Port Type: Analog (0-10V) ▼

Port Number: 1 ▼

Laser Position: -9.5 position at minimum 0 volts

Laser Position: 4.5 position at maximum 10 volts

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

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

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

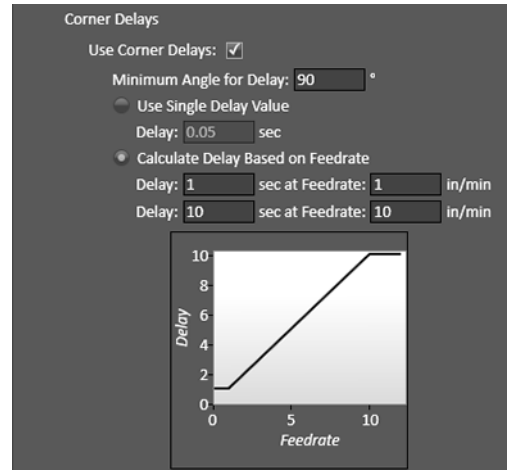
- **Corner Delays**

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

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

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

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



- **General Sensing**

Direction – Select direction.

Feedrate – Enter value in in/min.

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

General Sensing

Direction: ☐ Positive ☒ Negative

Feedrate: 10 in/min

Sensing Method: Monitor Input Line

Primary Sensor

Monitor Input Line: 19

Sensor Offset: 0 in

Use Additional Sensor: ☐

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

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

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

- **Program Zero Sensing**

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

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

Program Zero Sensing

Use Program Zero Sensing: ☒

Retract Distance: 0.25 in

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

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

Touch Off

Touch Off At Start (G605): ☒

Zero Lift Axis Program Coord: ☒

Rapid Move to Start Height: ☒

Enable Suppression Radius: ☒

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

Laser Pointer

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

The screenshot shows the 'General' settings panel. It includes fields for 'ID#' (set to 1), 'Name' (set to 'Laser Pointer 1'), 'Offset' with 'X' and 'Y' coordinates (both set to 0 in), and a 'Link To' dropdown menu (set to 'None').

Link To – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the drop down menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.

- **Laser Control**

Laser On/Off Output Lines –

When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

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

The screenshot shows the 'Laser Control' panel. It has two main sections: 'Laser On' and 'Laser Off'. Each section includes 'Output Lines' (set to 1), a status indicator (On/Off), a 'Custom Macro' checkbox (checked), and an 'Edit...' button. There are also '+' and 'x' icons for managing output lines.

- **Pointing**

Start/End of Point – Select these fields to define operations that take place at the start or end of a point. Selecting **Edit** will bring up the code editor window.

The screenshot shows the 'Pointing' panel. It includes two sections: 'Start of Point' and 'End of Point'. Each section has a checked checkbox, an 'Edit...' button, and a text area for defining the operation.

Marker

- **ID and Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended during automatic motion.
- **Link To** – Allows for the use of multiple simultaneous fabrication heads. Available heads will appear in the drop down menu. When selected, some settings will be greyed out because they are controlled by the linked Fab Head.
- **Marker Control**
 - **Marker On/Off Output Lines** – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.
 - **Custom Macro** – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting **Edit** will bring up the code editor window.
- **Marking**
 - **Start/End of Mark** – Select these fields to define operations that take place at the start or end of a mark. Selecting **Edit** will bring up the code editor window.

Type: Marker

General

ID#: 3

Name: Marker 3

Offset: X: 0 in Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Allow Runtime Jog: ☐

Link To: None

Marker Control

Marker On: Output Lines: [v] On [x] +

☒ Custom Macro Edit...

Marker Off: Output Lines: [v] Off [x] +

☒ Custom Macro Edit...

Marking

☒ Start of Mark: Edit...

☒ End of Mark: Edit...

- **General Sensing**

Direction – Select direction.

Feedrate – Enter value in in/min.

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

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

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

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

- **Program Zero Sensing**

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

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

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

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

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

- **Additional Controls**

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

Mill

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

Direction – Select direction.

Feedrate – Enter value in in/min.

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

If **Monitor Input Line** is selected, the following parameters apply:

- **Spindle**

Spindle Control – Select to enable.

Spindle Direction – Select Single- or Bi-Direction.

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

If Bi-Direction, Spindle On controls can be configured for clockwise and counter-clockwise (CW, CCW) operations.

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

Type: Mill

General

ID#: 1

Name: Spindle 1

Offset: X: 0 in, Y: 0 in

Spindle

Spindle Control ☒

Spindle Direction: Single Direction

Spindle On: Output Lines: 1 On

☒ Custom Macro Edit...

Spindle Off: Output Lines: 1 Off

☒ Custom Macro Edit...

Minimum Spindle Speed: 0

Maximum Spindle Speed: 3000

Control Spindle Speed: ☒

Port Type: Analog (0-10V)

Port Number: 1

Minimum rpm at Voltage: 0 volts

Maximum rpm at Voltage: 10 volts

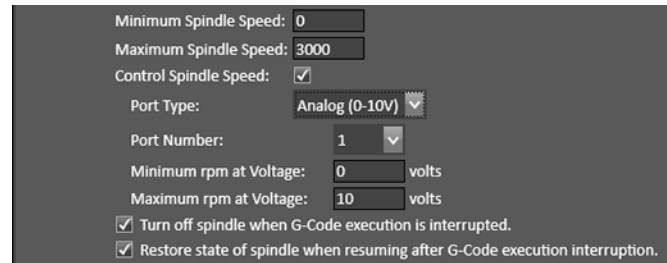
☒ Turn off spindle when G-Code execution is interrupted.

☒ Restore state of spindle when resuming after G-Code execution interruption.

Minimum/Maximum Spindle Speed – Enter value.

Control Spindle Speed – Select to enable

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



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

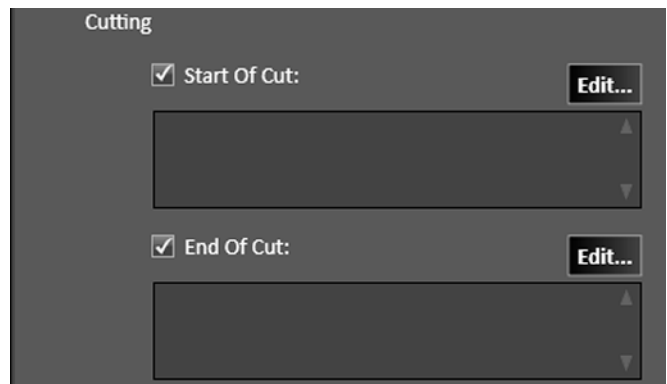
For **Analog**, select **Port Number**. For **PWM**, enter **Frequency**. For **PFM**, enter **Duty Cycle**.

Minimum/Maximum RPM at Voltage – Range varies for different port types.

G-Code execution interruption – Select to configure spindle behavior when G-Code is interrupted or when resuming.

- Cutting**

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



- Tool Length Sensing**

Use Tool Sensing – Select to enable.

Lift Axis Retract Position –

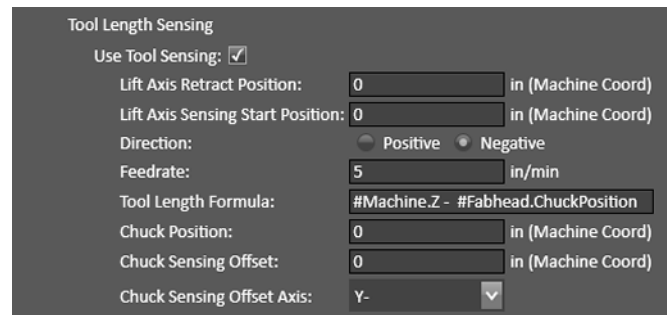
Lift Axis Sensing Start Position –

Direction – The direction of the move towards the sensor.

Feedrate – The feedrate for the move towards the sensor.

Tool Length Formula – The Z axis offset for the current tool is automatically entered into the Tool Library panel of the Configuration dialog box, according to the Tool Length Formula.

Chuck Position –



Chuck Sensing Offset –**Chuck Sensing Offset Axis** – Select axis, positive or negative.

- General Sensing**

Direction – Select direction.**Feedrate** – Enter value in in/min.**Sensing Method** – Select either **Monitor Input Line** or **Monitor Input Current**.**Sensor Offset** – The extra distance that the Z axis travels in the upwards direction after sensing the sheet surface. This distance can account for any hysteresis or play in the sensing switch or Z Axis mechanical system.**Use Additional Sensor** – A backup switch that trips during the touch off process in case the primary touch off method fails. This is typically a mechanical switch.

If Monitor Input current

If **Monitor Input Current** is selected, the following parameters apply:**Current Limit** – in amperes**Current Debounce** – in msec

- Program Zero Sensing**

Use Program Zero Sensing – Select to enable.**Retract Distance** – The amount that the Fab Head will move up after sensing the surface.

- Tool Change**

Sense Tool Length – Select to enable.

General Sensing

Direction: ☐ Positive ☒ Negative

Feedrate: in/min

Sensing Method: ▼

Primary Sensor

Monitor Input Line: ▼

Sensor Offset: in

Use Additional Sensor: ☒

Monitor Input Line: ▼

Sensor Offset: in

Sensing Method: ▼

Current Limit: A

Current Debounce: msec

Program Zero Sensing

Use Program Zero Sensing: ☒

Retract Distance: in

Tool Change

Sense Tool Length: ☒

- **Tool Changing**

Select **Silent**, **Manual**, or **Automatic**.

If **Automatic** is selected, the following parameters apply:

- **Monitor Chuck Power** – Select to enable.

Monitor Input Line – Select line.

Power On When Tripped – Select to enable.

The screenshot shows a 'Tool Changing' configuration window. It has three radio buttons: 'Silent (No Pause)', 'Manual (Pause With Message)', and 'Automatic' (which is selected). Below these are three checked checkboxes: 'Monitor Chuck Power', 'Monitor Input Line' (with a dropdown menu showing '10'), and 'Power On When Tripped'. There are two sections for 'Chuck Grip' and 'Chuck Release'. Each section has an 'Output Lines' label, a checked 'Custom Macro' checkbox, and an 'Edit...' button. To the right of each 'Output Lines' label is a '+' icon and a list box with up/down arrows.

Chuck Grip/Release Output Lines – When selected, define output line(s) used to control the Chuck. Add additional output lines by selecting the + icon; subtract output lines by selecting the x.

Custom Macro – Select these fields to define operations that take place when the Chuck Grip/Release is actuated. Selecting **Edit** will bring up the code editor window.

Tangential Knife

- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Lift Axis** – Select axis to be used when lifting the Fab Head.
- **Lift Axis Offset** – Enter value.
- **Rotary Axis** – Select axis.
- **Knife 0 Direction** – Select direction.

Rotary Axis Offset To 0 --

- **Oscillation**

Spindle Control – Select to enable.

Spindle Direction – Select Single- or Bi-Direction.

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

Control Oscillation Speed – Select to enable.

Minimum/Maximum

Frequency – Enter value in hz.

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

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

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

Minimum/Maximum RPM at Voltage – Range varies for different port types.

G-Code execution interruption – Select to configure spindle behavior when G-Code is interrupted or when resuming.

- **Cutting**

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

Cutting

Enable: ☒

☒ Start Of Cut: Edit...

☒ End Of Cut: Edit...

- **Marking**

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

Marking

Enable: ☒

☒ Start of Mark: Edit...

☒ End of Mark: Edit...

- **Tool Length Sensing**

Use Tool Sensing – Select to enable.

Lift Axis Retract Position –

Tool Length Sensing

Use Tool Sensing: ☒

Lift Axis Retract Position: 0 in (Machine Coord)

Lift Axis Sensing Start Position: 0 in (Machine Coord)

Direction: ☐ Positive ☒ Negative

Feedrate: 5 in/min

Tool Length Formula: #Machine.Z

Lift Axis Sensing Start Position –

Direction – The direction of the move towards the sensor.

Feedrate – The feedrate for the move towards the sensor.

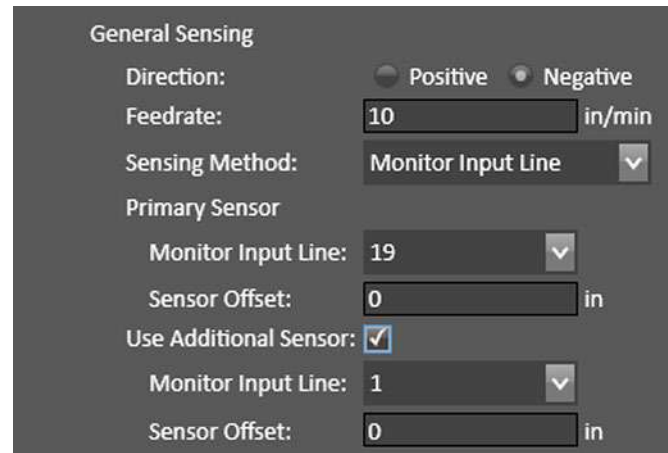
- **General Sensing**

Direction – Select direction.

Feedrate – Enter value in in/min.

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

Sensor Offset – The extra distance that the Z axis travels in the upwards direction after sensing.



General Sensing

Direction: ☐ Positive ☒ Negative

Feedrate: in/min

Sensing Method: ▼

Primary Sensor

Monitor Input Line: ▼

Sensor Offset: in

Use Additional Sensor: ☒

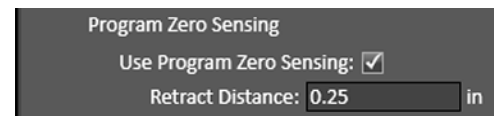
Monitor Input Line: ▼

Sensor Offset: in

- **Program Zero Sensing**

Use Program Zero Sensing – Select to enable.

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



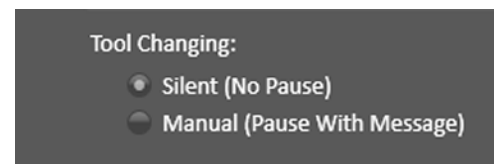
Program Zero Sensing

Use Program Zero Sensing: ☒

Retract Distance: in

- **Tool Changing**

Select **Silent** or **Manual**.



Tool Changing:

☒ Silent (No Pause)

☐ Manual (Pause With Message)

Hotwire

- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one Fab Head.
- **Hotwire Control** – When selected, define output line(s) used to control the Fab Head. Add additional output lines by selecting the **+** icon; subtract output lines by selecting the **x**.
- **Custom Macro** – Select these fields to define operations that take place when the Fab Head is turned on or off. Selecting **Edit** will bring up the code editor window.

The screenshot shows the 'Hotwire' configuration window. It is divided into two main sections: 'General' and 'Hotwire Control'.

General Section:

- ID#:** A text box containing the value '1'.
- Name:** A text box containing the value 'Hotwire'.
- Offset:** Two text boxes for 'X' and 'Y' offsets, both containing '0', followed by 'in'.

Hotwire Control Section:

- Enable:** A checkbox that is checked.
- Hotwire On:** A section with 'Output Lines: 1' (a dropdown menu), a green 'On' button with a close icon, and a '+' icon to add more lines.
- Custom Macro:** A checkbox that is checked, followed by an 'Edit...' button and a large text area for macro code.
- Hotwire Off:** A section with 'Output Lines: 1' (a dropdown menu), a grey 'Off' button with a close icon, and a '+' icon to add more lines.
- Custom Macro:** A checkbox that is checked, followed by an 'Edit...' button and a large text area for macro code.

Cut Charts

Cut charts are tables of information for configuring the properties of different manufacturer's Fab Heads. The information is stored in *.csv files in the FlashCut program files folder.

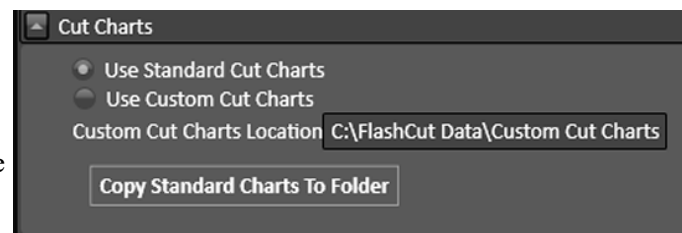
When configuring Fab Heads, the initial values for a particular head are populated from the relevant standard cut chart.

An example is shown below:

1	NozzleMc	TorchLeac	Material	Thickn	KerfWidth	CutHeight	PierceHei	PierceDel	BestFeed	BestVol	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																

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

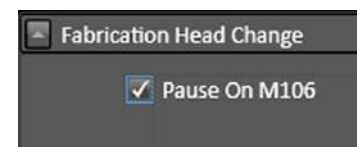
If you choose the Custom Cut Chart option, then a *.csv file copy of the standard cut charts will be stored into the specified location when you hit the **Copy Standard Charts To Folder** button.

Any value in these cut charts can be edited using a spreadsheet program like Excel.

Fabrication Head Change

Pause on M106

Select to enable.



Axes

- **Motor Line/Axis Mapping**
- **Axis Settings**
- **Feedrates and Ramping**
- **Drive Parameters**

Motor Line/Axis Mapping

Motor Line/Axis Mapping					
Line #	Axis		Sub Axis	Offset	Direction
1	Y	▼	1	▼	0 ☐ Positive ☒ Negative
2	X	▼	N/A	▼	0 ☐ Positive ☒ Negative
3	Z	▼	N/A	▼	0 ☒ Positive ☐ Negative
4	Y	▼	2	▼	0 ☐ Positive ☒ Negative
5	None	▼	N/A	▼	0 ☒ Positive ☐ Negative

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.

Note In general, axes X Y Z are always linear axes. Axes W V U could be linear or rotary depending on the specific machine. 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

Axis	Use Axis	Limited travel	Length (in)	Backlash (in)	Direction Change Delay (sec)	Interpolate During Safe Moves
X	☒	☒	48	0	0.01	☒
Y	☒	☒	48	0	0.01	☒
Z	☒	☐	6.5	0	0.01	☐
W	☐	☐	10	0	0.01	☐
V	☐	☐	10	0	0.01	☐
U	☐	☐	10	0	0.01	☐

☐ Enable Backlash Compensation

Enables you to define whether or not an axis is being used and if there are physical limits to each the axes 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

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/sec². 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 Rate 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).

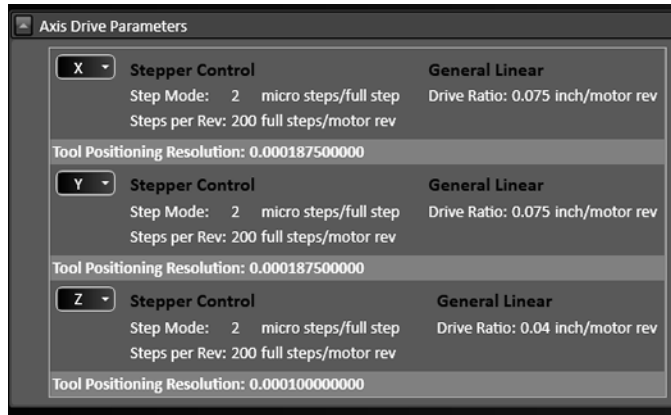
A typical default setting for the continuous contouring feedrate tolerance is the **Start/Stop Feedrate (in/min)** value. The higher you set the tolerance, the smoother a G-Code file will be executed (i.e., more moves will be linked together into smooth motions). If the values are set too high, loss 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.

Drive Parameters

Axis Drive Parameters

The axis drive parameters window displays the following information:

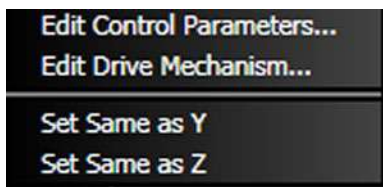


For each axis:

- 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 displays the following options (X Axis shown):



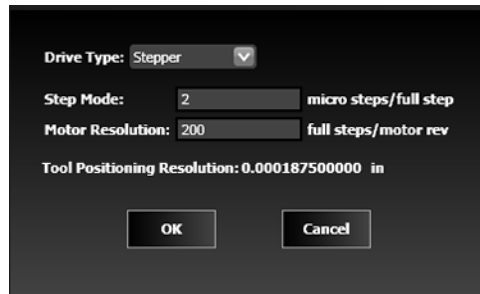
Set Same as

Sets all control/drive mechanism settings 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**

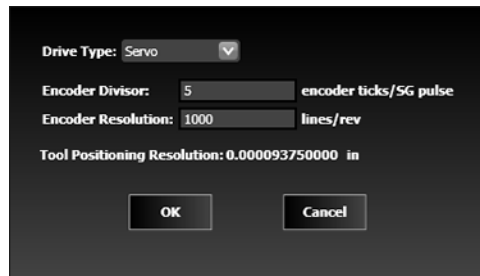


The screenshot shows the 'Stepper' configuration window. It features a 'Drive Type' dropdown menu set to 'Stepper'. Below it are three input fields: 'Step Mode' with a value of '2' and the unit 'micro steps/full step', 'Motor Resolution' with a value of '200' and the unit 'full steps/motor rev', and 'Tool Positioning Resolution' with a value of '0.000187500000 in'. At the bottom are 'OK' and 'Cancel' buttons.

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**



The screenshot shows the 'Servo' configuration window. It features a 'Drive Type' dropdown menu set to 'Servo'. Below it are three input fields: 'Encoder Divisor' with a value of '5' and the unit 'encoder ticks/SG pulse', 'Encoder Resolution' with a value of '1000' and the unit 'lines/rev', and 'Tool Positioning Resolution' with a value of '0.000093750000 in'. At the bottom are 'OK' and 'Cancel' buttons.

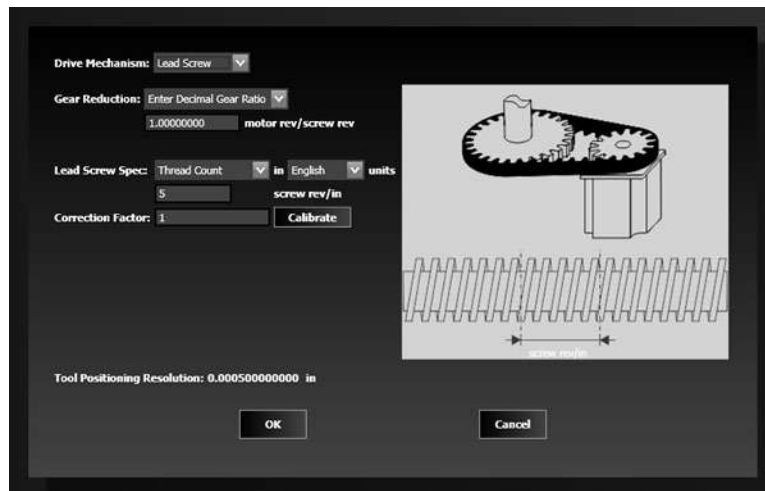
Encoder Divisor – The number of encoder pulses that correspond to one pulse from the Signal Generator.

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.

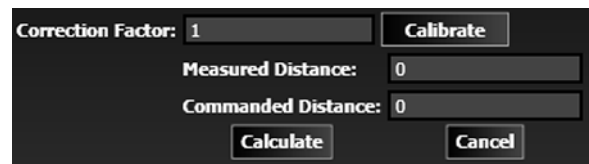
- Lead Screw**



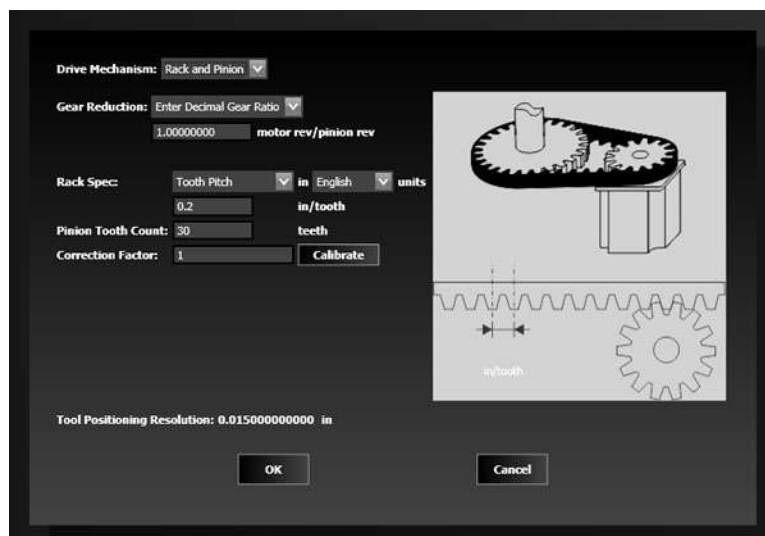
Gear Reduction – Enter the parameters for any gearing between the motor and lead screw. A decimal gear ratio of 1.0 indicates no gearing.

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**

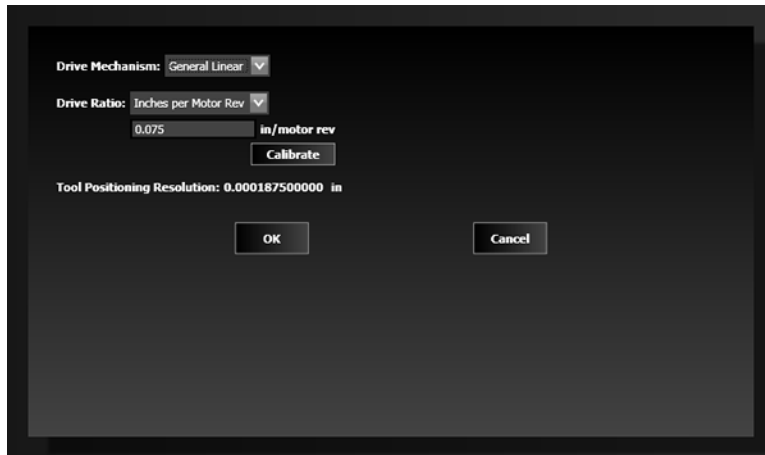
Gear Reduction – Enter the parameters for any gearing between the motor and drive pulley. A decimal gear ratio of 1.0 indicates no gearing.

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.

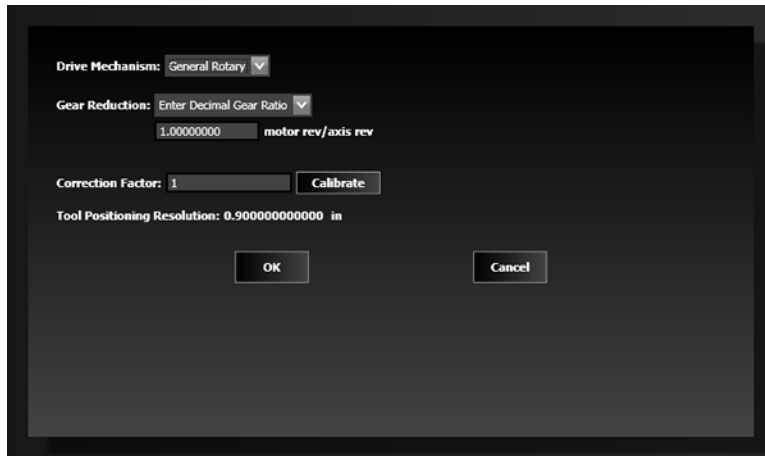
- **General Linear**

A screenshot of a software dialog box titled "General Linear". It contains a dropdown menu for "Drive Mechanism" set to "General Linear". Below it is a "Drive Ratio" section with a dropdown set to "Inches per Motor Rev" and a text input field containing "0.075". To the right of the input field is the unit "in/motor rev". Below the input field is a "Calibrate" button. At the bottom, it displays "Tool Positioning Resolution: 0.000187500000 in". At the very bottom are "OK" and "Cancel" buttons.

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

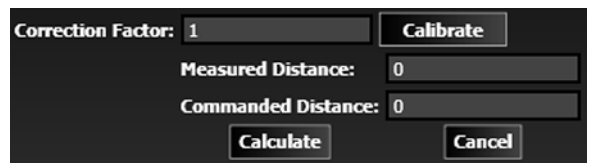
Calibrate – Display the Measured Distance and Commanded Distance fields. The operator commands the machine to move a specific distance, and then measures the actual distance travelled. The system then calculates the correction factor for the tool positioning resolution.

- **General Rotary**

A screenshot of a software dialog box titled "General Rotary". It contains a dropdown menu for "Drive Mechanism" set to "General Rotary". Below it is a "Gear Reduction" section with a dropdown set to "Enter Decimal Gear Ratio" and a text input field containing "1.00000000". To the right of the input field is the unit "motor rev/axis rev". Below the input field is a "Calibrate" button. At the bottom, it displays "Tool Positioning Resolution: 0.900000000000 in". At the very bottom are "OK" and "Cancel" buttons.

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.

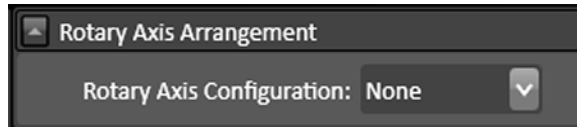
A screenshot of the calibration controls from the "General Rotary" dialog. It shows a "Correction Factor" section with a text input field containing "1" and a "Calibrate" button. Below this are two rows: "Measured Distance:" with a text input field containing "0", and "Commanded Distance:" with a text input field containing "0". At the bottom are "Calculate" and "Cancel" buttons.

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

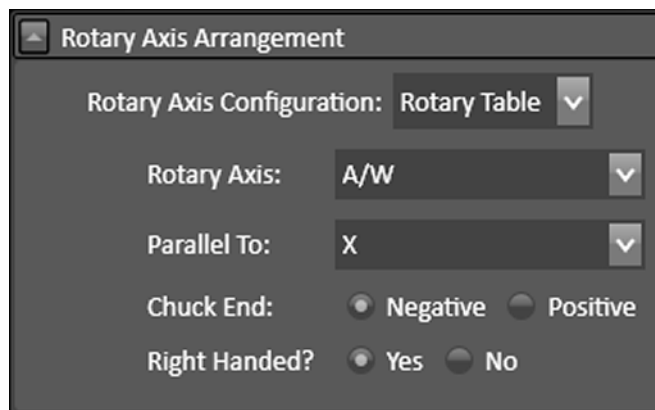
Select **None** if you are not using a rotary axis.

Other

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

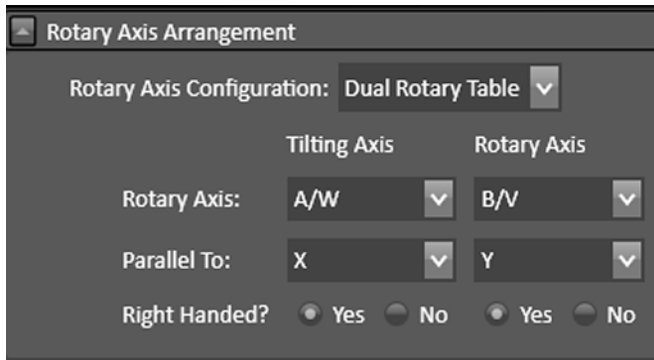
Rotary Table

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

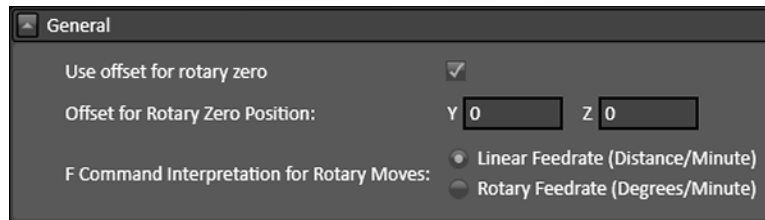
Select Dual Rotary Table if you are using two rotary axes built into a single table. You must define a Rotary Axis for each.



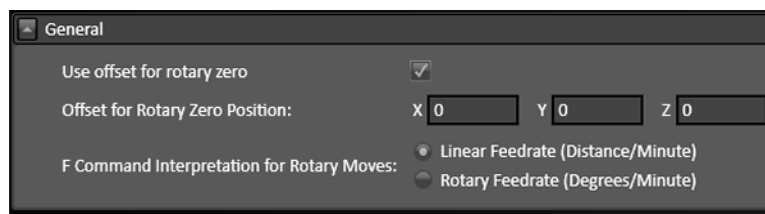
General

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.

Reference Points								
G-Code	Description	Coord System	X Coord (in)	Y Coord (in)	Z Coord (in)	W Coord (in)	V Coord (in)	U Coord (in)
G28	Tool Change	Machine ▼	0	0	0	0	0	0
G30 P2		Machine ▼	0	0	0	0	0	0
G30 P3		Machine ▼	0	0	0	0	0	0
G30 P4		Machine ▼	0	0	0	0	0	0
G30 P5		Machine ▼	0	0	0	0	0	0
G30 P6		Machine ▼	0	0	0	0	0	0
G30 P7		Machine ▼	0	0	0	0	0	0
G30 P8		Machine ▼	0	0	0	0	0	0
G30 P9		Machine ▼	0	0	0	0	0	0
G30 P10		Machine ▼	0	0	0	0	0	0
G30 P11		Machine ▼	0	0	0	0	0	0
G30 P12		Machine ▼	0	0	0	0	0	0
G30 P13		Machine ▼	0	0	0	0	0	0
G30 P14		Machine ▼	0	0	0	0	0	0
G30 P15		Machine ▼	0	0	0	0	0	0
G30 P16		Machine ▼	0	0	0	0	0	0
G30 P17		Machine ▼	0	0	0	0	0	0
G30 P18		Machine ▼	0	0	0	0	0	0
G30 P19		Machine ▼	0	0	0	0	0	0
G30 P20		Machine ▼	0	0	0	0	0	0

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.

Fixture Offsets								
G-Code	Description	X Offset (in)	Y Offset (in)	Z Offset (in)	W Offset (in)	V Offset (in)	C Offset (deg)	
G54	Preset 1	0	0	-3	0	0	0	
G55	Preset 2	0	0	0	0	0	0	
G56	Preset 3	0	0	0	0	0	0	
G57	Preset 4	0	0	0	0	0	0	
G58	Preset 5	0	0	0	0	0	0	
G59	Preset 6	0	0	0	0	0	0	

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:

Axis	Home Axis	Home End	Home Order	Home Switch Offset	Homing Rate	Homing Tolerance	Execute M-Code Before	Execute M-Code After	Home Position Machine Coordinate
X	☒	☐ Positive ☒ Negative	1 v	0.5 in	40 in /min	0 in			0 in
Y	☒	☐ Positive ☒ Negative	2 v	0.5 in	40 in /min	0 in			0 in
Z	☐	☒ Positive ☐ Negative	3 v	0.1 in	10 in /min	0 in			6 in
W	☐	☐ Positive ☒ Negative	4 v	0.1 in	10 in /min	0 in			0 in
V	☐	☐ Positive ☒ Negative	5 v	0.1 in	10 in /min	0 in			0 in
U	☐	☐ Positive ☒ Negative	6 v	0.1 in	10 in /min	0 in			0 in

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

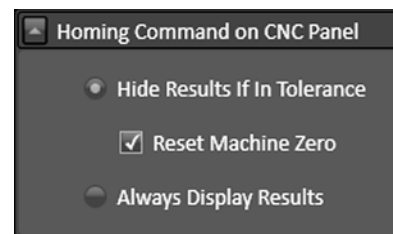
If the home switch is positioned at the travel limit, the machine coordinate of the home switch is zero. If the home switch is positioned somewhere within the machine envelope, set this value to the machine coordinate of the home switch location (nonzero value).

Homing Command on CNC Panel

These settings apply to the homing process initiated by the Homing Command on the CNC Panel.

Hide Results if in Tolerance

Tells FlashCut not to display the results of re-homing, if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.



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.

Always Display Results

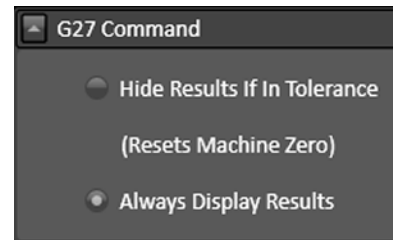
Tells FlashCut to always display the results of re-homing, even if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

G27 Command

These settings apply to the homing process initiated by the G27 Command. They work the same as the Seek Home Button controls described above, except that if you choose Hide Results if in Tolerance, FlashCut always resets Machine Zero to the new location found.

Hide Results if in Tolerance

Tells FlashCut not to display the results of re-homing, if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.



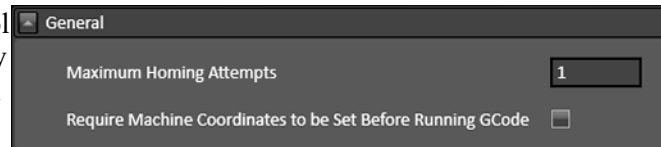
Always Display Results

Tells FlashCut to always display the results of re-homing, even if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

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.

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.

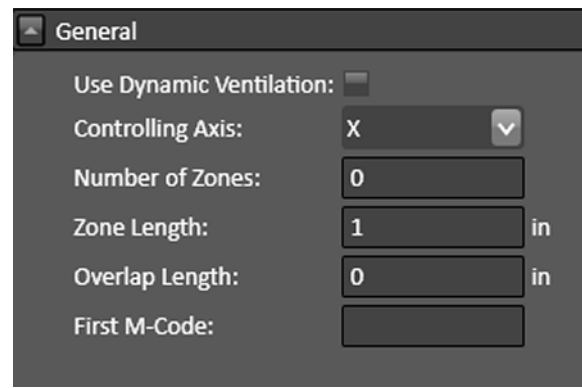
General

Use Dynamic Ventilation

When checked, the dynamic ventilation system will be active.

Controlling Axis

Sets the axis along which the ventilation zones are defined.



The screenshot shows a 'General' settings window with the following controls:

- Use Dynamic Ventilation:** A checkbox that is currently checked.
- Controlling Axis:** A dropdown menu set to 'X'.
- Number of Zones:** A text input field containing the value '0'.
- Zone Length:** A text input field containing the value '1', followed by a unit selector set to 'in'.
- Overlap Length:** A text input field containing the value '0', followed by a unit selector set to 'in'.
- First M-Code:** An empty text input field.

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

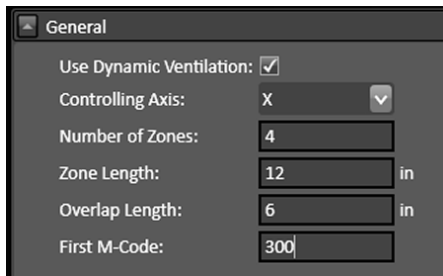
Enter the first M-Code here. The first M-Code is the code that opens the damper in zone one. Note that **codes are consecutive**: the second M-Code will close the damper for zone one, and so on.

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:



The screenshot shows the 'General' configuration panel for Dynamic Ventilation. It contains the following settings:

Parameter	Value	Unit
Use Dynamic Ventilation:	☒	
Controlling Axis:	X	
Number of Zones:	4	
Zone Length:	12	in
Overlap Length:	6	in
First M-Code:	300	

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 Fab Head settings for **Mill**.

Tool Changer

Tool Rack Style

Select Fixed, Carousel, or Custom

Loading Type

Select Top or Side Loading

Approach Feedrate

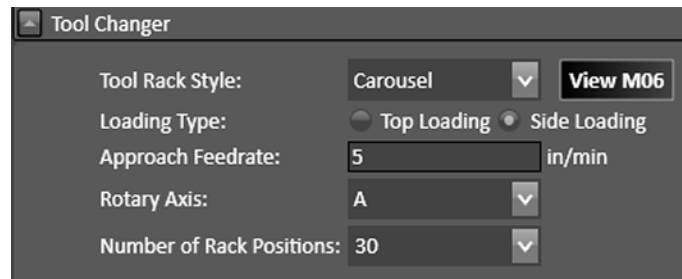
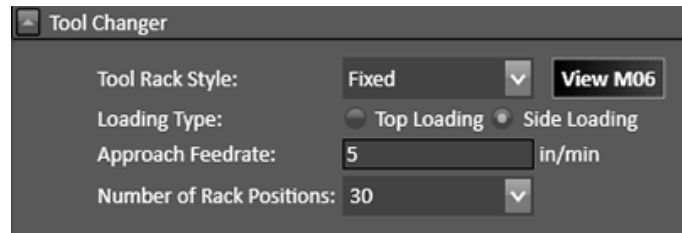
Feedrate for all moves to the Lift Axis Grip and Release positions

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

Tool Rack Positions		Units: Inches							
Position	X Axis Position	Y Axis Position	Loading Safe Position X	Loading Safe Position Y	Lift Axis Grip	Lift Axis Release	Lift Axis Near Grip	Lift Axis Near Release	Lift Axis Raised
1	0	0	0	0	0	0	0	0	0
2	0	0	0	0	0	0	0	0	0
3	0	0	0	0	0	0	0	0	0

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.

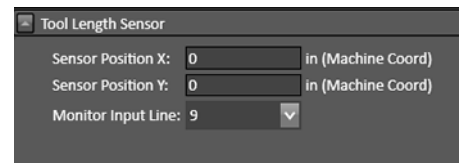
WARNING: IT IS VERY IMPORTANT TO SET THIS VALUE HIGH ENOUGH TO ACCOMMODATE THE LONGEST TOOL. OTHERWISE A CRASH WILL OCCUR.

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. For more parameters, see Fab Head settings for **Mill**.



The screenshot shows a software panel titled "Tool Length Sensor". It contains three input fields: "Sensor Position X:" with a value of "0", "Sensor Position Y:" with a value of "0", and "Monitor Input Line:" with a value of "9". Each of the first two fields has a small "in (Machine Coord)" label to its right. The "Monitor Input Line:" field has a dropdown arrow on its right side.

Programming

- **M-Code Definitions**
- **M-Code Execution**
- **G-Code**

M-Code Definitions

The M-Code definition option presents these M-Code definition configuration types:

- **Output Line M-Codes**
- **Macro M-Codes**

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.

Output Line M-Codes					
M-Code	Description	Execute Before Move	Execute Delay First	Delay (sec)	
50	Torch On	☒	☐	0.5	1 ▼ On X
51	Torch Off	☒	☐	0	1 ▼ Off X

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.

Macro M-Codes			
M-Code	Description	Macro	Execute Before Move
80	Start ATHC PM65	M41	☐
81	Stop ATHC PM65	G600	☐
82	Start ATHC PM45XP Air	M43	☐
83	Stop ATHC PM45XP	G600	☐
84	Start ATHC PM45XP Arg	M42	☐

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

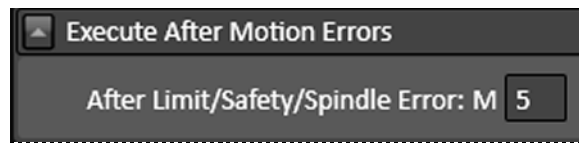
The M-Code execution option presents these M-Code execution configuration types:

- **Execute After Motion Errors**
- **Execute at Start/End of Motion**
- **Execute During Ramping**
- **Execute While Connecting/Disconnecting**

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

Execute After Motion Errors

- **After Limit/Safety/Spindle Error** – FlashCut executes the M-Code after an error occurs. This is useful in applications where it's safest to automatically turn off the spindle, vacuum and other devices whenever the signal generator detects an error.



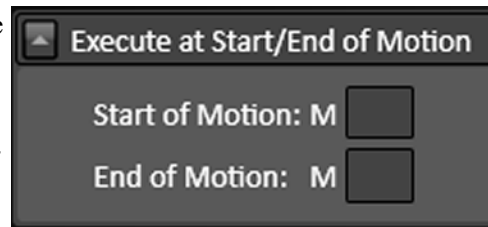
Errors include limit switch tripped, safety switch tripped, or spindle stopped during thread cutting (lathe only).

When running a lathe, and especially when cutting threads, this mode is highly recommended. If an error occurs (e.g. a limit switch gets tripped), FlashCut immediately stops moving the tool. The spindle must then be stopped, or else there will be interference between the stationary tool and the spinning workpiece. The automatic M-Code should be configured to stop the spindle.

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 During Ramping

A typical use for this feature is to turn on/off an external device when the machine tool is accelerating or decelerating. FlashCut turns the device on/off only while running a G-Code file.

- **Beginning of Ramp Up** – FlashCut executes this M-Code when ramping up begins.

Execute During Ramping		
Beginning of Ramp Up:	M	☐
End of Ramp Up:	M	☐
Beginning of Ramp Down:	M	☐
End of Ramp Down:	M	☐

- **End of Ramp Up** – FlashCut executes this M-Code when ramping up ends.
- **Beginning of Ramp Down** – FlashCut executes this M-Code when ramping down begins.
- **End of Ramp Down** – FlashCut executes this M-Code when ramping down ends.

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.

Execute While Connecting/Disconnecting		
Connecting:	M	☐
Disconnecting:	M	☐

G-Code

General	
Arc Radius Tolerance:	0.01 in
G02/G03 Arc Center Parameters (I/J/K):	Incremental (Distance from Start) Absolute (Program Coordinate)
Performance	
Preprocess G-Code During File Load:	☒
Preview Toolpath in Viewports:	☒
Calculate Run Time for Progress Meter:	☐
Optional Commands	
Execute G54 - G59:	☒
Buffer Sizes	
Loading:	100000
Buffering:	10000
Reverse:	10000
Timing:	10000

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.

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

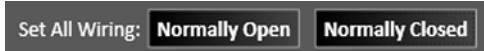
- **Input Lines**
- **Output Lines**

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**
- **Description**
- **Function**
- **Axis**
- **Sub-axis**
- **Position**
- **Sensing Debounce**
- **Wiring**



Line #	Description	Function	Axis	Sub Axis	Position	Sensing Debounce	Wiring
1		Unused ▼	N/A ▼	N/A ▼	N/A ▼	0.001	☐ N.O. ☒ N.C.
2		Unused ▼	N/A ▼	N/A ▼	N/A ▼	0.001	☐ N.O. ☒ N.C.

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).

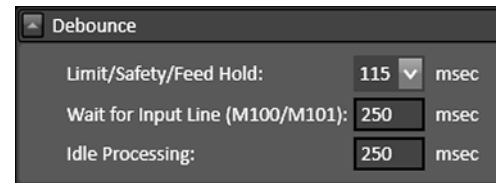
Wiring

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

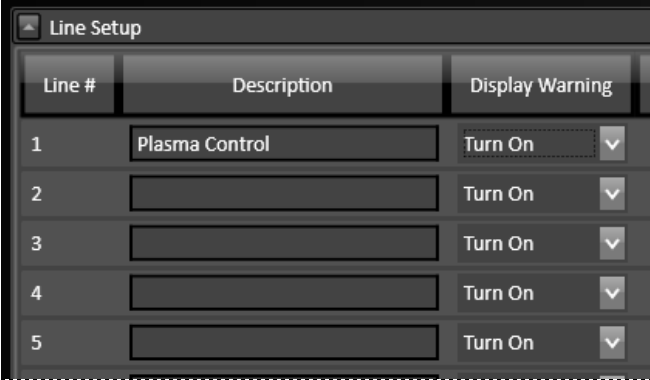
This is the duration that a switch must be continuously tripped for FlashCut to recognize it as tripped.

- **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**.



Line #	Description	Display Warning
1	Plasma Control	Turn On
2		Turn On
3		Turn On
4		Turn On
5		Turn On

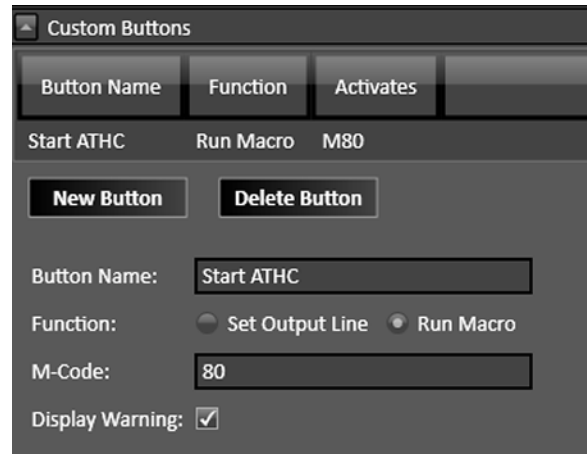
User Interface

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).

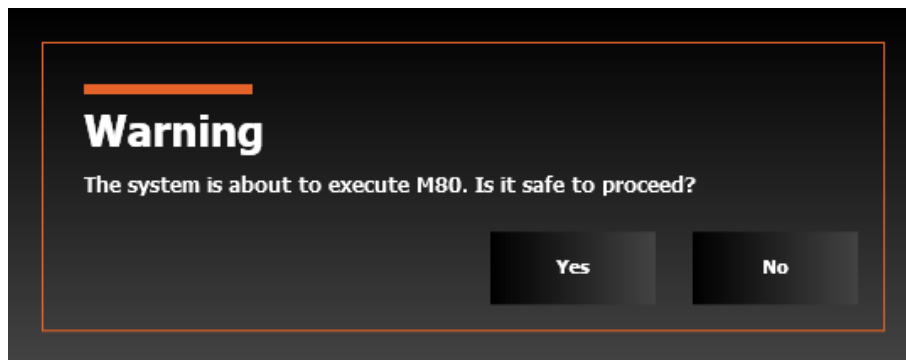
Note that the M-Code has already been defined, and that the Display Warning checkbox has been selected.



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.



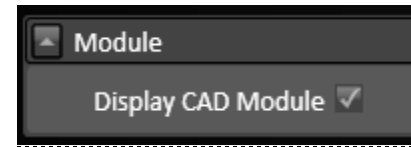
CAD

- General
- Features

General

Display CAD Module

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



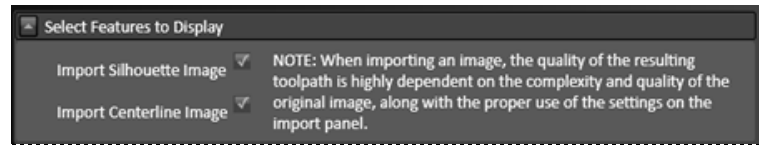
Features

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.

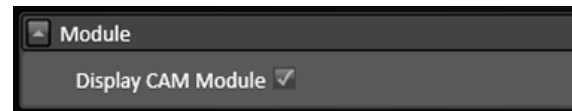
CAM

- General
- Features
- Post Processor

General

Module

When selected the CAM module will be available in FlashCut.



Geometry Processing

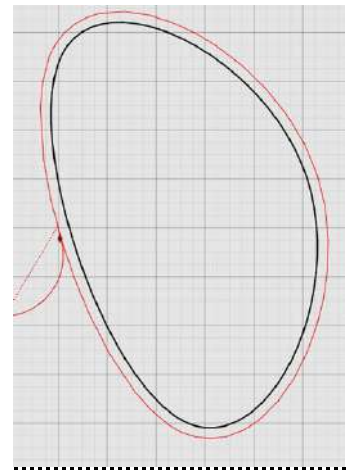
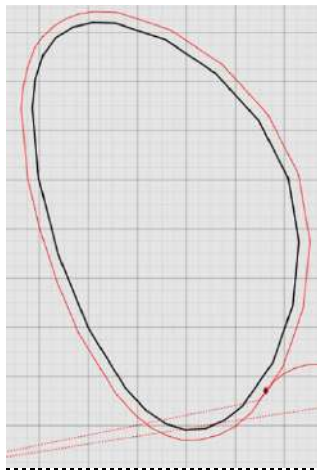
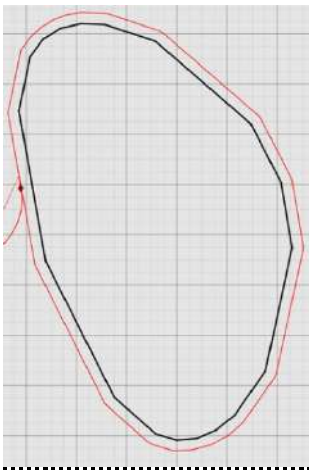
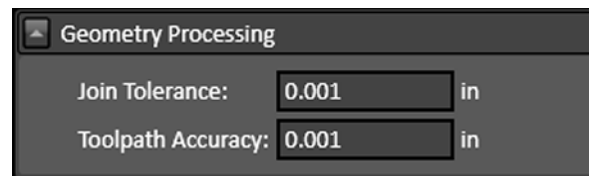
- **Join tolerance**

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).

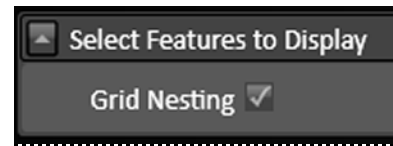


Features

- **Select Features to Display**

Grid Nesting

This enables the use of grid nesting in the CAM module.



Post Processor

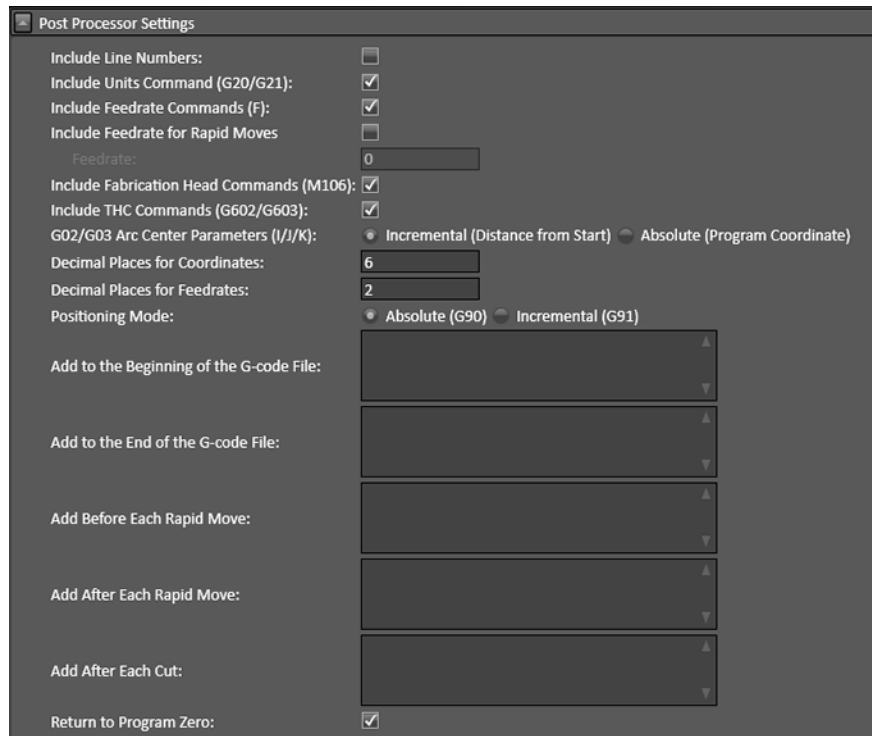
Post Processor Settings

Include line numbers

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

Include units command

Check this box if you want to include the G20/G21 commands for Inch/Metric units respectively. The G-Code command will correspond with the units defined in the CAD/CAM settings menu.



Include feedrate commands

Keep this box enabled: disabling this box is intended for internal use only. The feedrate command will no longer be included in the program if this box is disabled.

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

Include feedrate for rapid moves

This will enable the use of the **F** feedrate command for a rapid move. Select the checkbox and enter the feedrate value to be used in the input field labeled **feedrate**.

Include fabrication head commands

If selected, the M-106 command will be included in the G-Code file. M-106 is the Fab Head change command: for example, changing from plasma Fab Head to the marker Fab Head.

Include THC commands

For plasma fabrication heads only. Commands to enable and disable torch height control will be included in the G-Code file.

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

Controls the definition of clockwise (G02) and counterclockwise (G03) circular arcs. Select either incremental (distance from start), or absolute (program coordinates).

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.

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.

Activate

Enable the **Activate** checkbox to upload your G-Code file to an FTP server each time each time the user clicks the cut button.

A screenshot of a software dialog box titled "Save G-Code to FTP Server". The dialog has a dark gray background. It contains four labeled fields: "Activate" with a checked checkbox, "Server Address and Path:" with a wide text input field, "Username:" with a medium text input field, and "Password:" with a medium text input field. At the bottom, there is a label "Use FTP upload panel:" followed by a checked checkbox.

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.

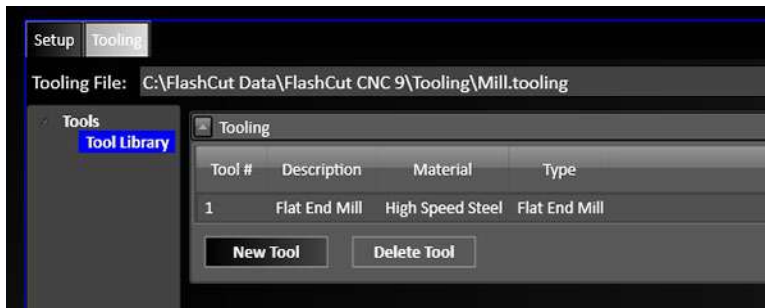
Use FTP upload panel

If **Use FTP upload panel** is checked: An FTP upload panel appears in the parameters pane with various control options.

If **Use FTP upload panel** is unchecked: the file is uploaded with a default filename (e.g., gcode.nc).

Tooling Configuration Tab

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

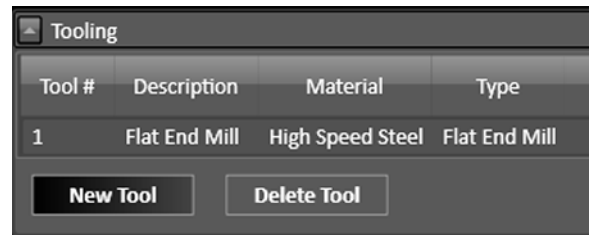


Tool Library

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 **System Units**. This definition is independent of the General **System Units** setting.

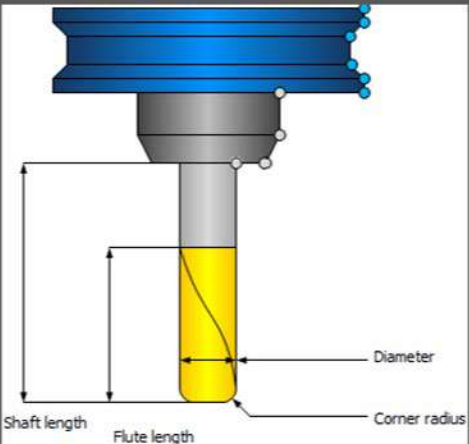
Sample tool configurations are shown on the following pages.

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°

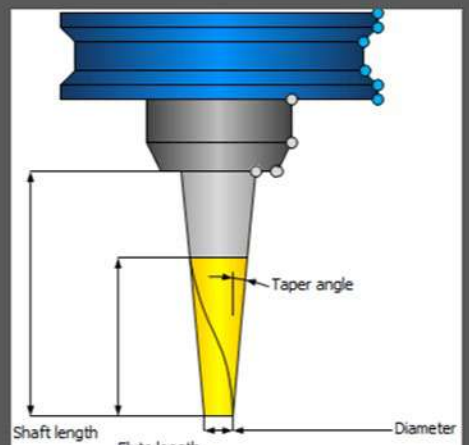
Tool Number:	2
Tool Offset:	0 in
System Units:	English
Description:	Ball End Mill
Material:	Uncoated Carbide
Nominal Diameter:	0.5000 in
Flute Length:	2.0000 in
Shaft Length:	4.0000 in
# Of Flutes:	2
Cutting Direction:	CW

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



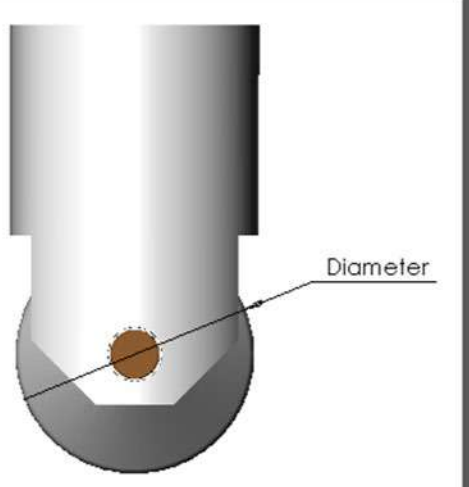
The diagram shows a side view of a round edge end mill. The tool has a blue cutting edge with four flutes. The shaft is yellow. Labels indicate the 'Shaft length' (total length), 'Flute length' (length of the fluted section), 'Diameter' (nominal diameter), and 'Corner radius' (at the bottom of the shaft).

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°



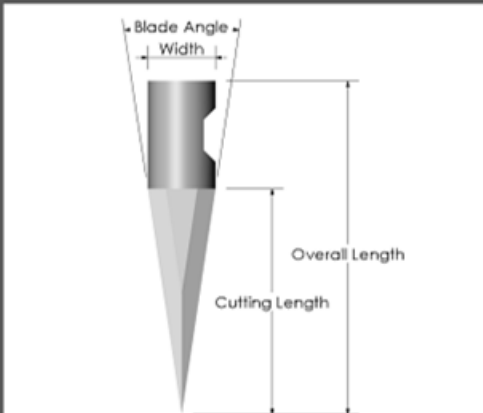
The diagram shows a side view of a tapered edge end mill. The tool has a blue cutting edge with three flutes. The shaft is yellow and tapers to a point. Labels indicate the 'Shaft length' (total length), 'Flute length' (length of the fluted section), 'Diameter' (at the base), and 'Taper angle' (the angle of the taper).

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

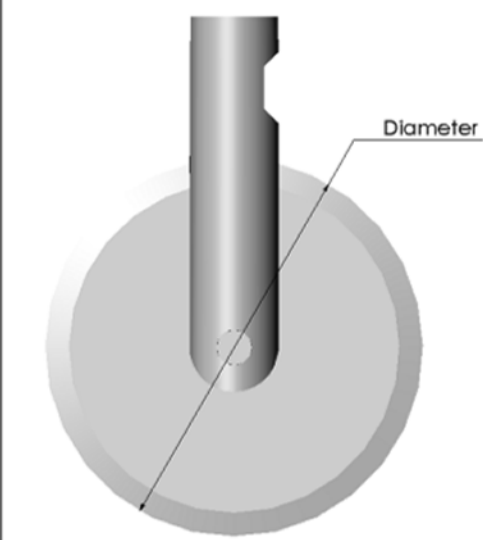


The diagram shows a side view of a creasing wheel. The tool has a grey cylindrical body with a brown, textured circular creasing wheel at the bottom. A label indicates the 'Diameter' of the main body.

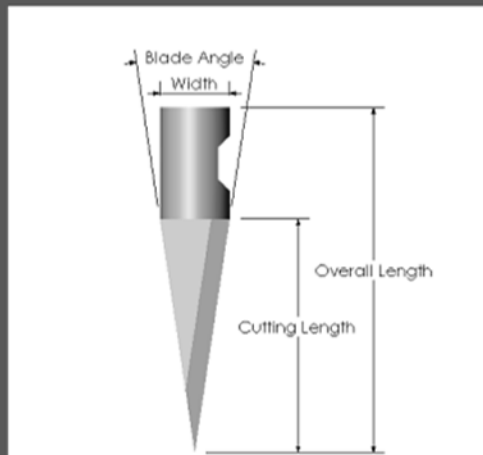
Tool Number:	8
Tool Offset:	0 in
System Units:	English
Description:	Dual Sided Knife
Cutting Length:	2 in
Width:	0.25 in
Blade Angle:	7.16 °
Overall Length:	4 in



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



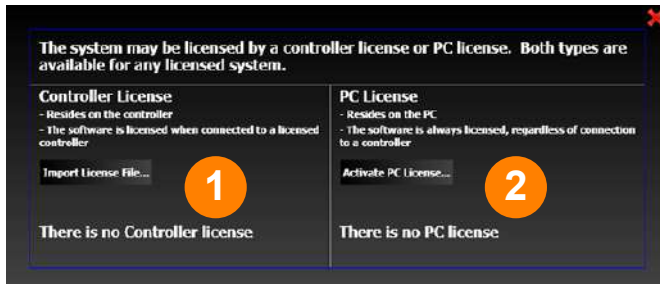
Licensing FlashCut



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

There are two forms of licensing in FlashCut: a license file for the controller, and a license key for the computer connected to the controller. Both licenses can be managed through FlashCut. A list of features enabled by the currently installed license can also be reviewed in this window.

Note An internet connection may be required when upgrading between versions to activate an existing PC license.



1

Import License File

2

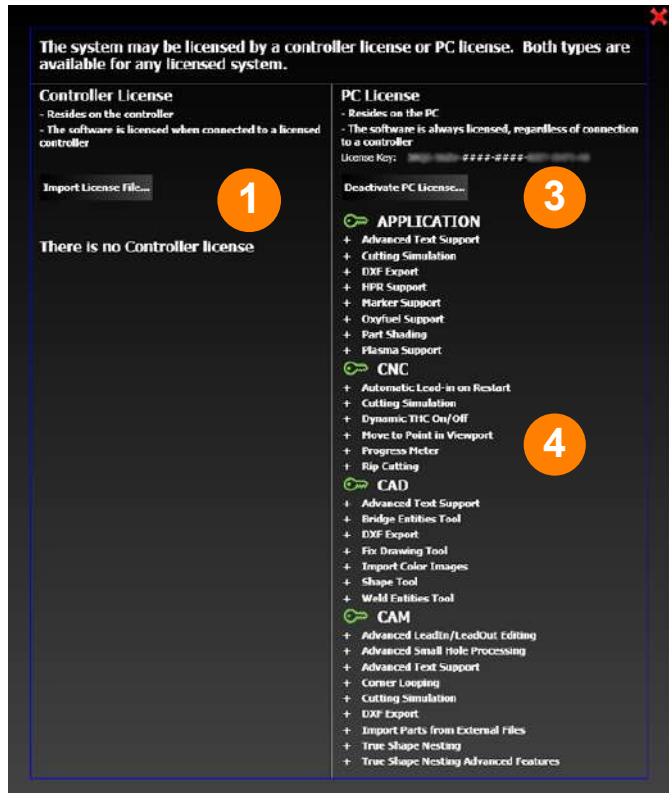
Activate PC License

3

Deactivate PC License

4

Licensed Features List



License Management Window

Import License File

If you've received a license file, you may import the file into this software to license it. Follow these steps to import your license into your software.

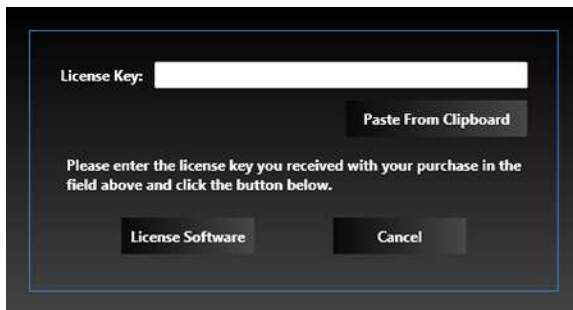
1. When you receive a license file, copy it to an appropriate location on your computer.
2. Click the **Import License File...** button.
3. FlashCut displays an **Open** dialog box.
4. Browse to the location of the file, select it, and click **OK**.

5. FlashCut installs the license and displays this notice to confirm that the license has been activated.
6. Click **OK**. The license will be loaded into your controller the next time you connect to it.

Activate PC License

If you've received a license key in an email message, you may license your software by typing or pasting the license key into a dialog box. Follow these steps to license your software with a license key from an email message.

1. Click the **Activate PC License...** button.
2. FlashCut displays a **License Your Software** dialog box.

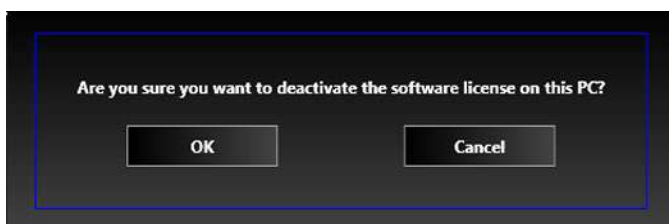


3. If you've copied the license key from the email you received, click the **Paste From Clipboard** button. FlashCut pastes the license key in the **License Key** field. Otherwise, you may type the license key in the field yourself.
4. Click the **License Software** button.
5. FlashCut installs the license and displays a notice to confirm that the license has been activated. The license will be loaded into your controller the next time you connect to it.
6. The license will be loaded into your controller the next time you connect to it.

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.

7. Click the **Deactivate PC License...** button.
8. FlashCut displays a confirmation dialog box.



9. 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 FlashCut features, see **Program packages and features**.

Getting Help

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.

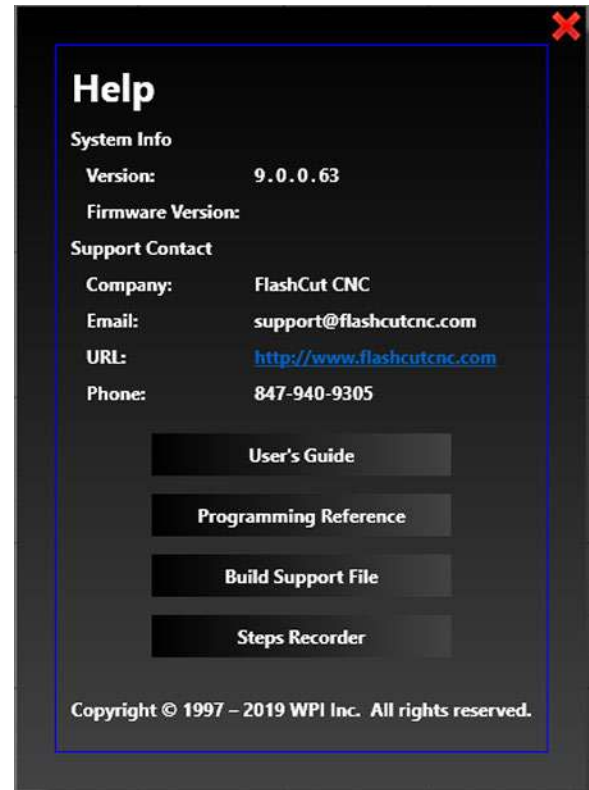
- **Version**
- **Firmware Version**
- **Support Contact**
- **User's Guide**
- **Programming Reference**
- **Build Support File**
- **Build Support File**

Version

The software revision identification number that is currently installed on the PC.

Firmware Version

The firmware revision identification number that is currently loaded on the signal generator.



Support Contact

Displays the support contact information for your version of the software.

User's Guide

Displays this User's Guide in PDF format. Requires a PDF viewer, such as Adobe® Acrobat®

Programming Reference

Displays the FlashCut Programming Reference in PDF format. Requires a PDF viewer, such as Adobe® Acrobat®

Build Support File

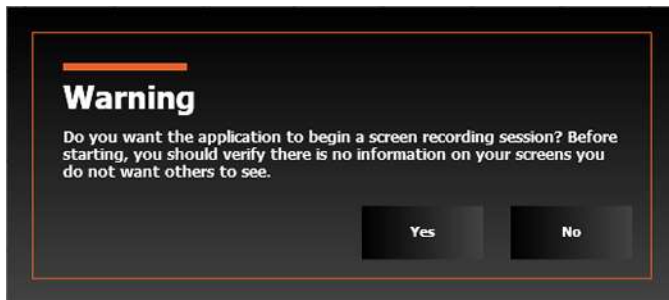
Creates a support file. A support file is a compressed ZIP file that contains all current information about your system, including copies of current setup, configuration, and log files. This information can help FlashCut technical support answer a question or diagnose a problem.

The loaded G-Code file will be included **only if it has been saved**. The support file **will not** include any CAD/CAM files. If you are having a CAD/CAM issue, please e-mail the CAD/CAM file separately.

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

Begins a screen recording session. 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. Select this icon to **Stop Recording**.

