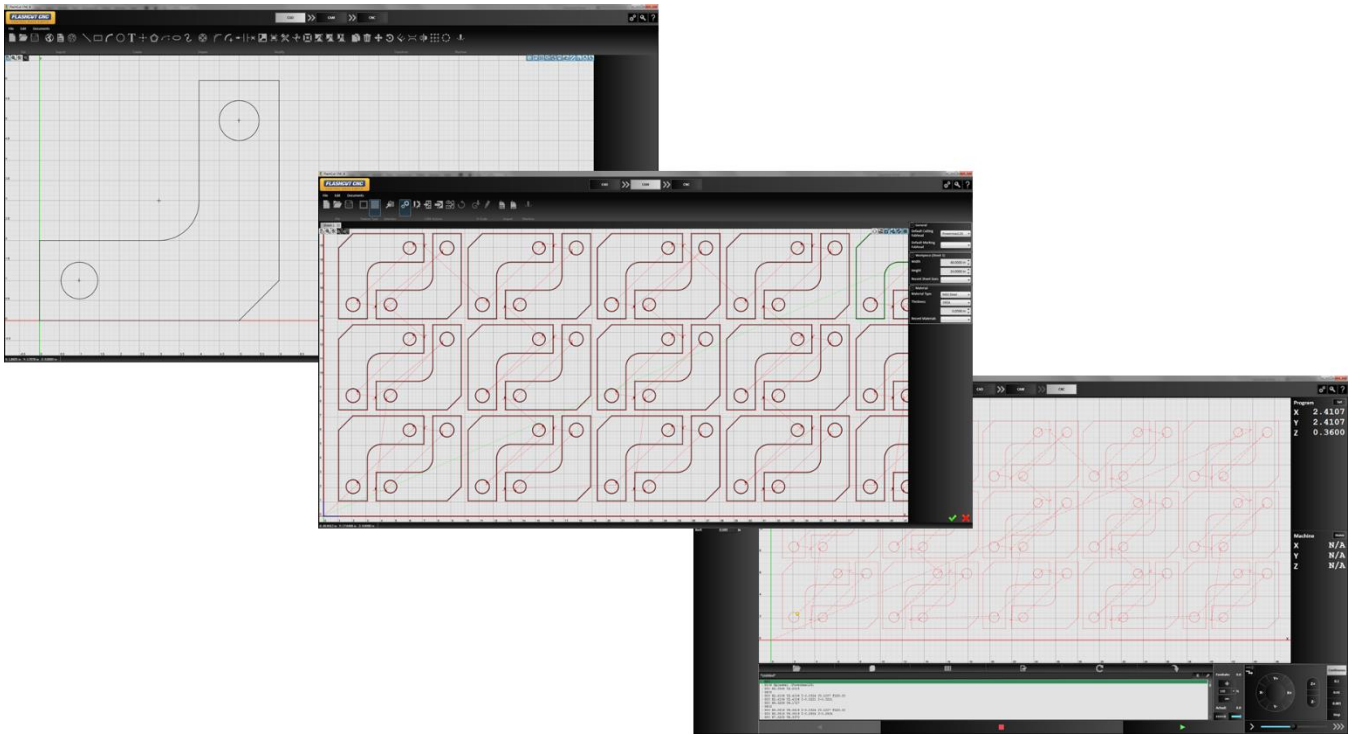
FLASHCUT CNC

CONTROL MADE SIMPLE ®

FlashCut CAD/CAM and CNC Control Software



Notices

Revised January 20, 2017
Version 6.0.6.100
© 1997-2016 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.
Other trademarks are the property of their respective holders.

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

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

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

Note The Support file will only include a G-Code file that has been saved and it **will not** include a CADCAM file. If you are having a CADCAM issue, please e-mail the CADCAM 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 1
- System requirements 1
- Program packages and features 1
  - Version 6 standard features 1
  - New CAD features for Version 6 2
  - New CAM features for Version 6 2
  - New CNC features for Version 6 3
  - Version 6 features available a la carte 3
  - Version comparison 3
- Installing the FlashCut CNC software 5
  - Preparing to install the software 5
  - Installing the software 5
  - Updating the setup file 8
- Installing the USB driver 9
  - Windows 7 9
  - Windows 8 and 10 13

Getting started with FlashCut CNC 14
- Launching the software 14
  - Running in evaluation mode 14
- Activating a license 14
- Configuring the software 14
- Getting help 15
- Understanding common interface elements 15
  - Configuration, License, and Help buttons 15
  - Tabs 15
  - Accept / Cancel 16
  - Menu bar 16
  - Documents menu 20
  - Parameters area 21
  - Status bar 21

FlashCut CAD 22
- Ribbon 23
  - File tools 23
  - Import tools 24
  - Create tools 31
  - Shape tool 44
  - Modify tools 45
  - Group 53
  - Ungroup 54
  - Ungroup All 54
  - Transform tools 55
- Pan and zoom tools 64
  - Pan 64
  - Zoom Extent 64

Zoom 64
Zoom to Fit 64
Snap tools 65

FlashCut CAM 69

Ribbon 70
File tools 70
Feature Type 70
Selection 71
CAM Actions 73
G-Code 84
Import 84
DXF/DWG 84
CADCAM 84
Pan and zoom tools 85
Pan 85
Zoom Workpiece Extents 85
Zoom 85
Zoom Toolpath Extents 85
Zoom Window 85
Display options tools 86
Cutting Directions 86
Kerf 86
Material 86
Parts 86
Rapid Moves 86
Tool Paths 86
Drawing workspace 87
Feature manipulation 87
Lead in/out settings 87
Program zero 89

FlashCut CNC 90

System Status, Toggle Display Units, and Connect 91
System Status 91
Toggle Display Units 91
Connect 92
Fab head settings 92
Laser pointer control 92
Plasma torch settings 92
Oxyfuel settings 94
Waterjet settings 94
Marker control 94
Program coordinates panel 95
Coordinate rotation example 96
Enter rotation angle 96
Use 2-point alignment 96
Machine Coordinates Panel 98
Jog and point control panel 99
Move to point 99
Jogging 100

G-Code window 100
Pan and zoom tools 103
Drawing workspace 104

Configuring FlashCut 105
Setup file (*.setup extension) 106
Configuration workspace 106
Configuration Menu 107
System 108
General 108
Controller 108
Preferences 113
Language 113
Jogging 113
Display 115
Pendant 116
USB Interface Board 117
Machine 117
Fabrication Heads 117
Axes 127
Drive Parameters 132
Rotary Axes 136
Reference Points 139
Homing 140
Dynamic Ventilation 142
Dynamic Ventilation Example 144
Programming 145
M-Code Definitions 145
M-Code Execution 146
I/O 149
Input Lines 149
CAD 151
General 151
Features 151
CAM 152
General 152
Features 153
Breaks 153
Post Processor 157
Toolpath Adjustments 160

Licensing FlashCut 161
License Management Window 162
Import License File 162
Activate PC License 162
Deactivate PC License 163
Licensed Features List 163

Getting Help 164
Help 164
Version 164
Firmware Version 164

Graphics Drivers 164
User's Guide 164
Programming Reference 165
Build Support File 165

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 6 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 (32 or 64 Bit)
- Intel I3 processor (or equivalent), or better
- At least 4GB of RAM
- 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. Other features add advanced functionality, and offer enhanced machine control during production. These advanced features can be purchased a la carte, and a pro package that includes the most popular a la carte features is also available.

Version 6 standard features

- Simple, powerful 2-D CAD
- Integrated CAM with true shape nesting
- Automatic kerf compensation
- Automatic generation of lead ins and lead outs

- Automatic cut settings based on material
- Simple import of pictures and drawings
- Easy CNC program flow control including jump to line, forward, and reverse processing
- Integrated Plasma Torch Height Control (THC)
- Industry renowned CNC control

New CAD features for Version 6

Standard Package

- Improved DXF/DWG import
- Measuring tool lets you determine any part dimension or distance on a drawing

Pro Package

- Integrated shape library
- DXF export
- Ability to bring True-Type fonts directly into CAM without exploding
- "Fix Tool" finds and corrects CAD mistakes instantly
- Ability to bridge internal cutouts instantly, such as the center of an "O"

New CAM features for Version 6

Standard Package

- Improved nesting performance
- Editable plasma and oxyfuel cut charts for Hypertherm® and Thermal Dynamics®
- Support for multiple fabrication heads including plasma, waterjet, oxyfuel, and spindle
- Smart feedrate and THC handling of small holes

Pro Package

- Enhanced nesting interface of multiple parts and external files
- Enhance production nesting features – fill sheet, multi sheet, partial sheet
- Corner looping

New CNC features for Version 6

Standard Package

- Marker support
- Advanced security to prohibit unwanted changes to configuration
- Support for simultaneous multiple fabrication heads including plasma, waterjet, oxyfuel, and spindle
- Feedhold on loss of Arc Transfer Signal
- CNC display units can be changed: inches or mm
- Support for linear and rotary axes
- Improvements to controls and cutting parameters for oxyfuel and waterjet fabrication heads

Pro Package

- Lead in automatically added when starting in the middle of a G-Code file, minimizing blemishes on part
- Ability to turn THC on and off in the middle of a cut

Version 6 features available a la carte

- Laser pointer support, including finding zero and previewing the toolpath directly on the sheet
- Enhanced communication with Hypertherm® Powermax® plasma cutters to set amperage, pressure and reporting status
- Bar code input and automatic file loading for streamlined production
- Advanced input configuration including joystick jogging
- Dynamic control of ventilation zones
- Coordinate rotation for workpiece alignment

Version comparison

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

Version 6.0 Feature Summary	Standard Package	Pro Package	Available A la carte
Simple, powerful 2-D CAD with improved DXF/DWG import	x	x	
Measuring tool displays any part dimension or distance	x	x	
Integrated CAM with true shape nesting with improved performance	x	x	
Editable plasma and oxyfuel cut charts for Hypertherm® and Thermal Dynamics®	x	x	
Support for multiple fabrication heads including plasma, waterjet, oxyfuel, and spindle	x	x	
Smart feed rate and Torch Height Control (THC) processing for small holes	x	x	
Automatic kerf compensation	x	x	
Automatic generation of lead ins and lead outs	x	x	
Automatic cut settings based on material	x	x	
Simple import of pictures and drawings	x	x	
Easy CNC program flow control including jump to line, forward, and reverse processing	x	x	
Integrated shape library		x	x
DXF export		x	x
Ability to bring True-Type fonts directly into CAM without exploding		x	x
Dynamically curve text around any shape including arcs, lines, ellipses and splines		x	x
Fix tool finds and corrects CAD mistakes instantly		x	x
Ability to bridge internal cutouts instantly (e.g., alphabet shapes)		x	x
One click to weld or subtract intersecting shapes		x	x
Enhanced nesting interface of multiple parts and external files		x	x
Enhance production nesting features - fill sheet, multi sheet, partial sheet		x	x
Corner looping		x	x
Ability to turn THC on and off while cutting		x	x
Automatic lead in on restart		x	x
Advanced file management including bar code scanning support and automatic file loading			x
Laser pointer and laser pointer software support *			x
Laser pointer software support only			x
Dynamic ventilation zones			x
Coordinate rotation for workpiece alignment			x
Programmable I/O interface board*			x
Joystick with Programmable I/O Interface Board *			x
Integrated THC with Mini I/O Expansion Board, Voltage Isolation Box, Ohmic and Torch Interface Cables *			x
RS-485 communications with Hypertherm® Powermax® plasmas for real-time current and pressure control *			x
* 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

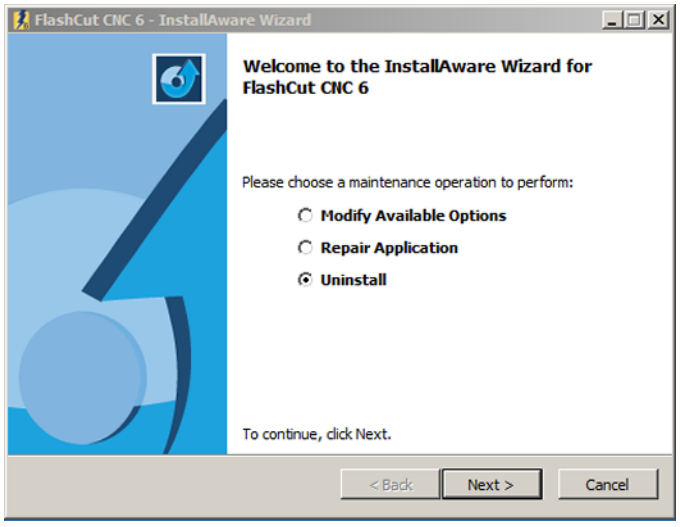
Please follow these instructions to install FlashCut CNC.

1. Open the FlashCut CNC installer downloaded from the internet, or mount the installation CD.

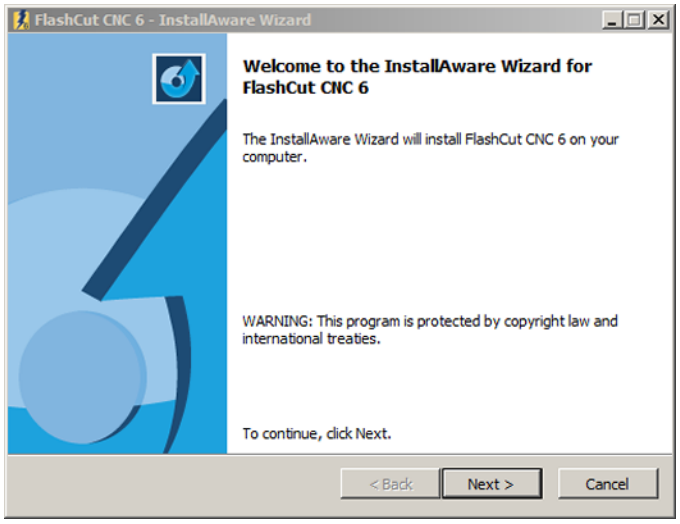
2. When prompted to run the installer, select **Run** to open the InstallAware wizard.

3. The wizard checks for previous installations of FlashCut CNC version 5 or 6, and prompts you to uninstall them.

Note You may also uninstall a version of FlashCut CNC 6 manually by selecting **Uninstall FlashCut CNC 6** from the Windows **Start** menu.

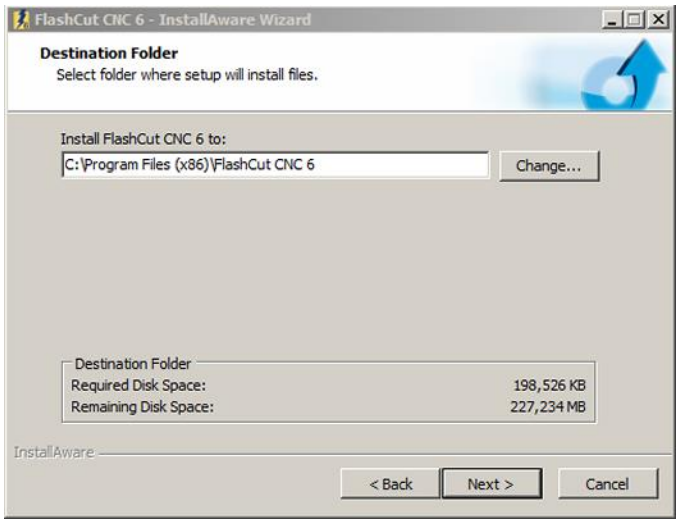


4. After dealing with the pre-installation requirements, the wizard prompts you to install the latest release.



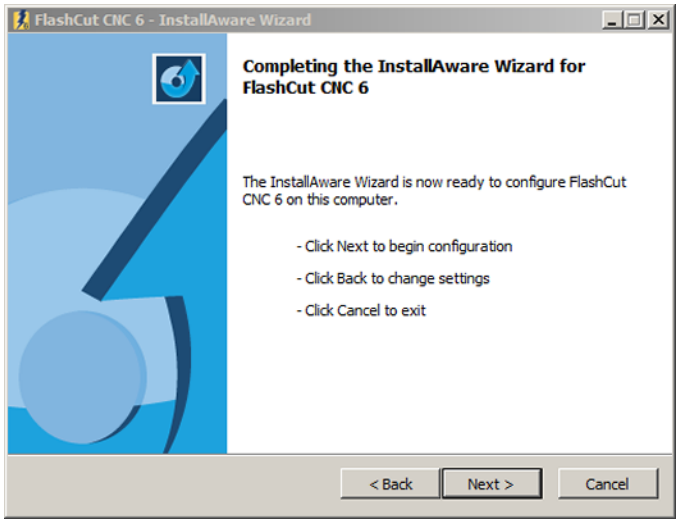
Click **Next >**.

5. 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 >**.

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

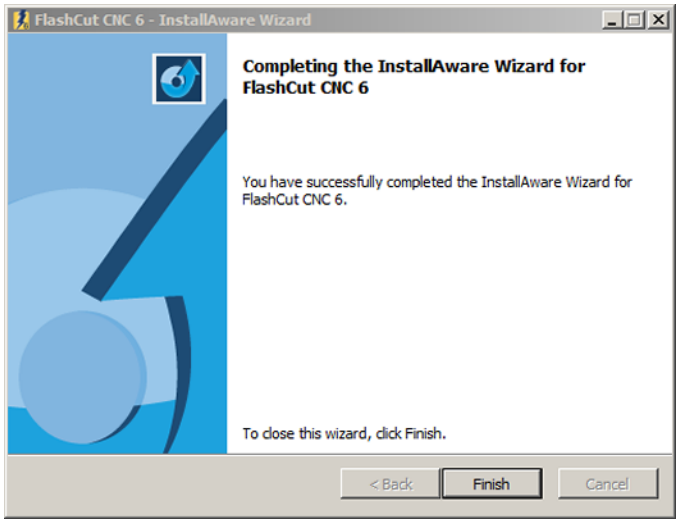


Click **Next >** to finalize the configuration and begin the installation.

7. 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.

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

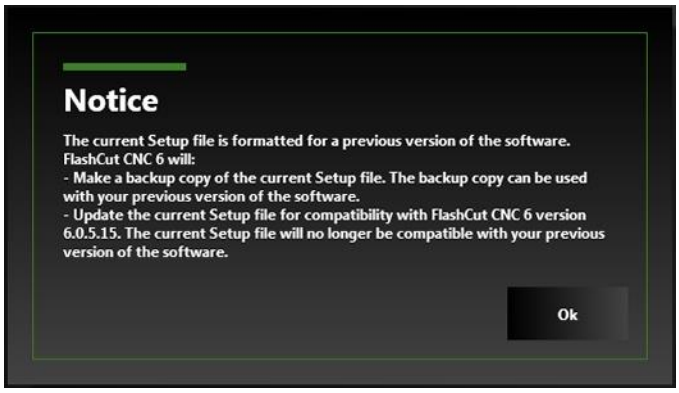


Click **Finish** to close the wizard.

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 Setup file (*.setup extension) in 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.

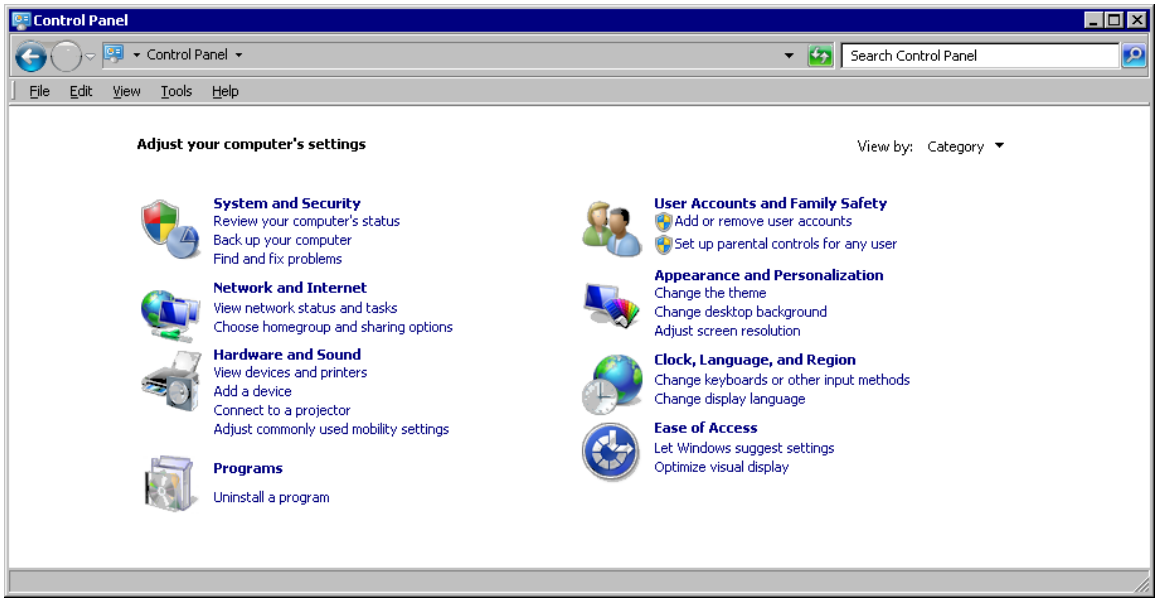
1. Make sure the FlashCut software is installed.
2. Connect the signal generator to the PC using an A-B USB cable.
3. Turn on the signal generator or CNC controller.

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

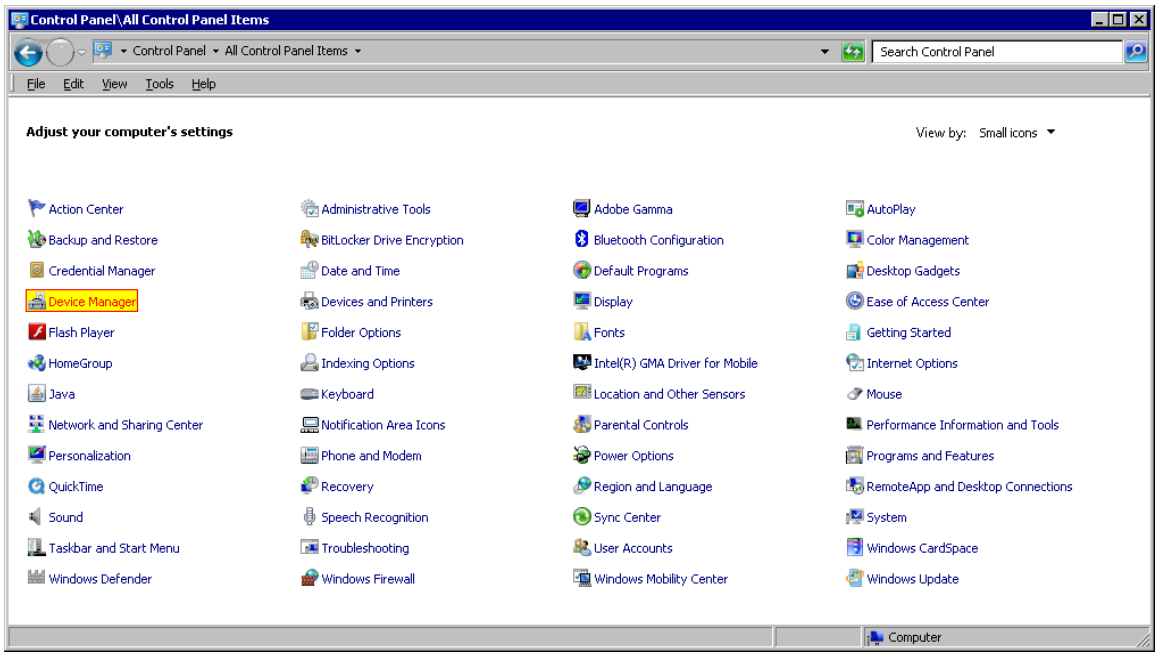
Windows 7

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.

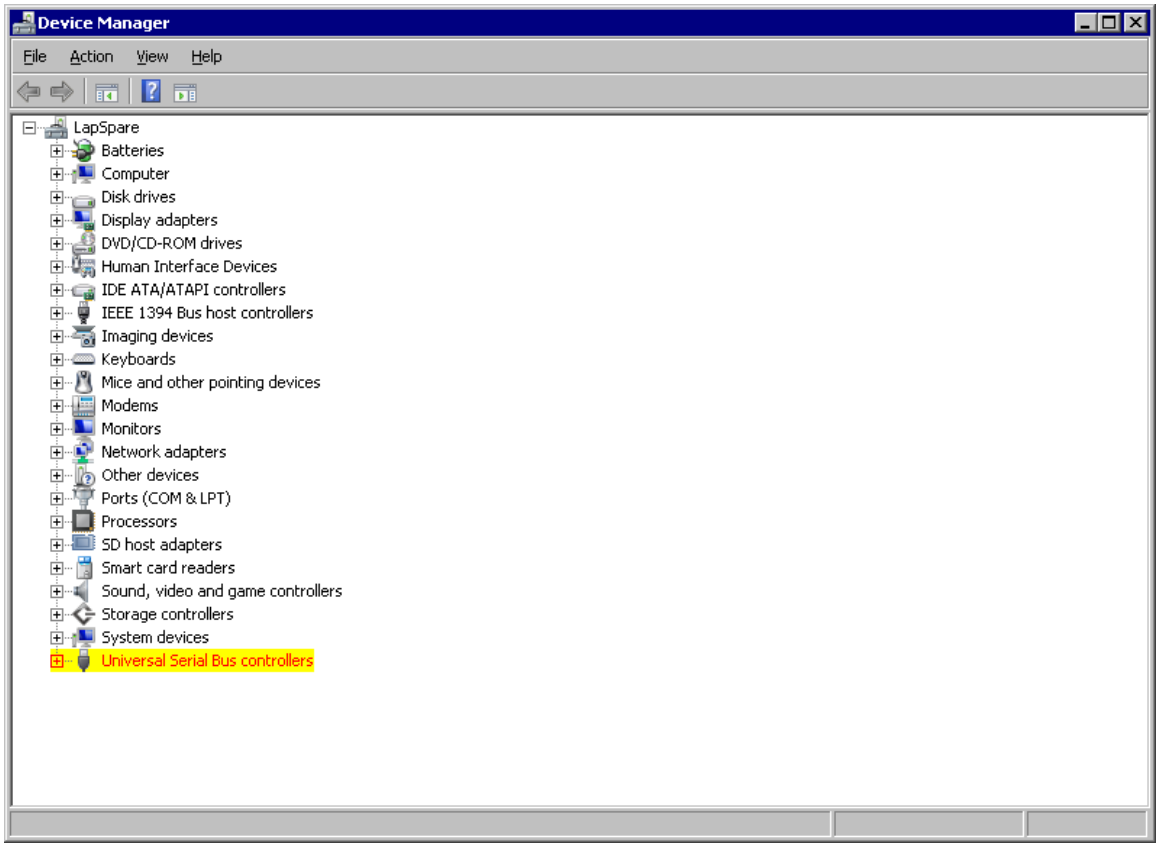
1. Click **Start►Control Panel**. Windows displays the Control Panel screen.



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.

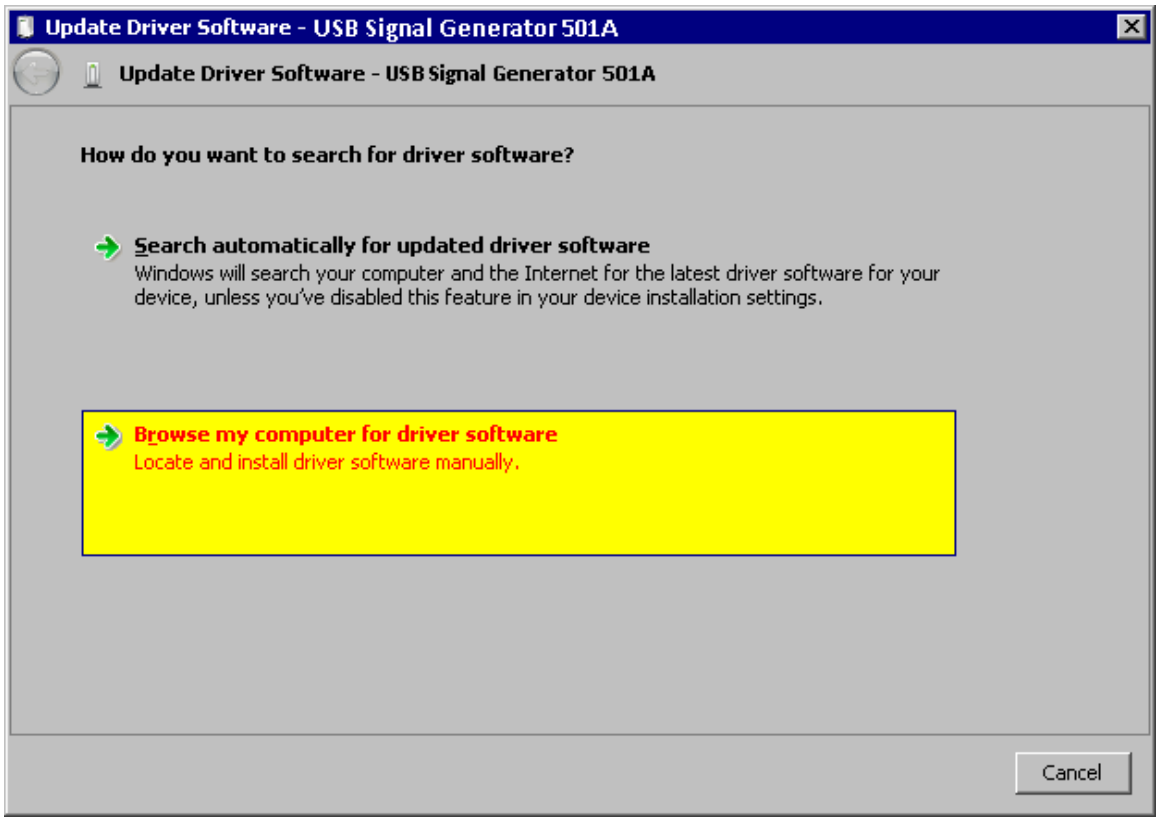


3. Click **Device Manager**. Windows displays the Device Manager screen.



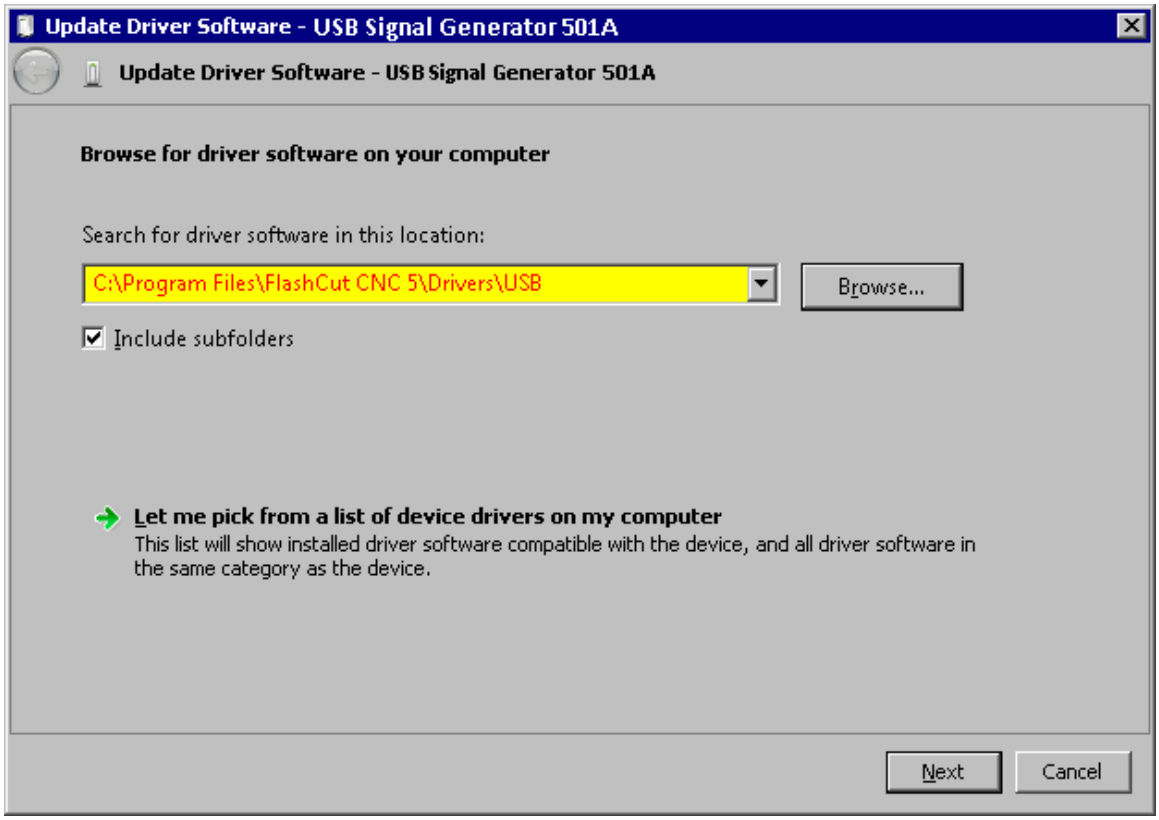
4. Expand the **Universal Serial Bus Controllers** listing.

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:

- **32-bit Windows 7**

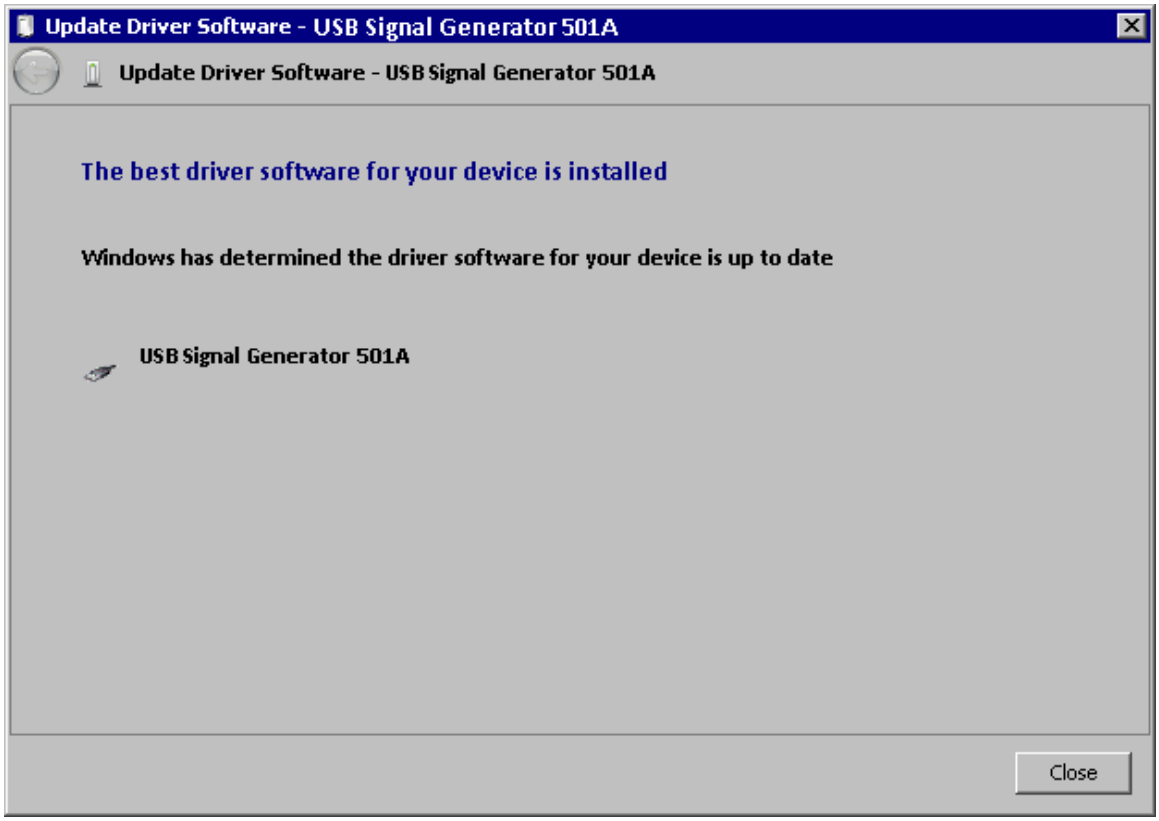
  C:\Program Files\FlashCut CNC 6\Drivers\USB

- **64-bit Windows 7**

  C:\Program Files (x86)\FlashCut CNC 6\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**.

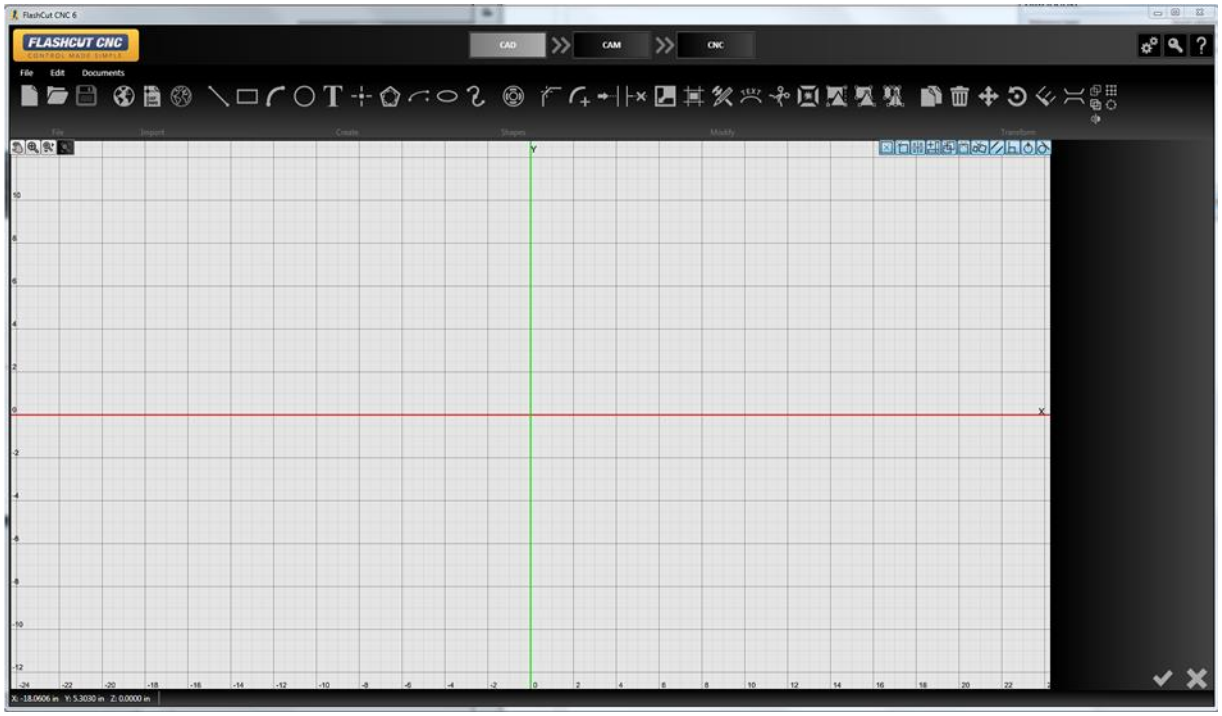
Windows 8 and 10

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

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 6**

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. See **Configuring FlashCut** for more information.

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

The **Help** interface is used to access this manual and the Programming Reference, and to generate 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

New (Ctrl+N)

Creates a new CAD/CAM drawing in a new FlashCut CAD window. It 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 such as 5.0.

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 6 or later. The Save command is unavailable when no recent changes have been made.

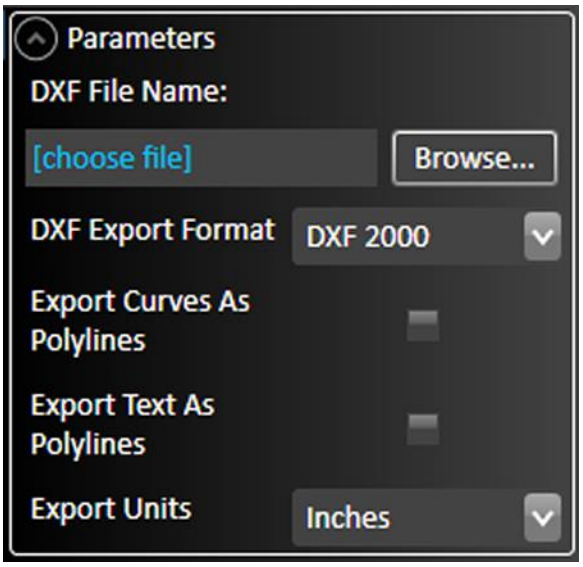
Save as

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 6 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 DXF**

  Saves a copy of the current drawing to a new file in the AutoCAD® Drawing Interchange Format (DXF).

  After setting the parameters for the file, you may either **Accept** or **Cancel** the changes.

- **Choose DXF file**

  Create a new file or replace an existing one. Browse opens the Windows dialog box.

- **DXF Export Format**

  Choose from DXF 2000 or DXF R12

- **Export Curves as Polylines**

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

- **Export Text as Polylines**

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

- **Export Units**

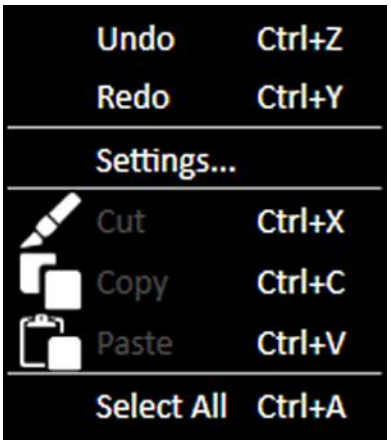
  Choose from either inches or millimeters

Exit

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

Edit menu

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



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

Undo (Ctrl+Z)

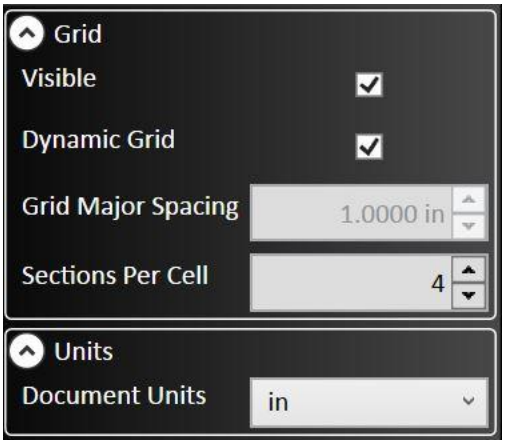
Reverses the previous drawing action. Up to 20 actions can be reversed.

Redo (Ctrl+Y)

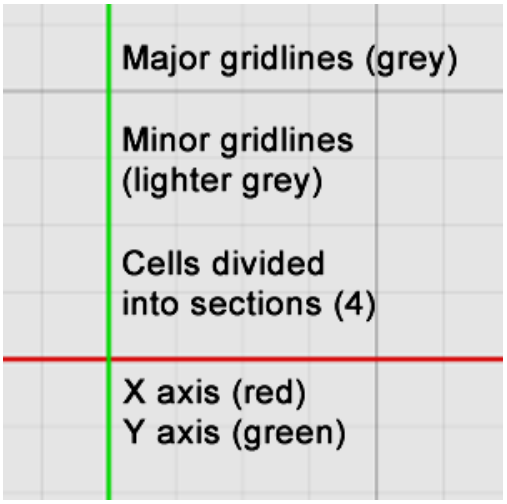
Repeats the previous drawing action, or reverses the Undo stack.

Settings...

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



Grid



- **Visible**

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

- **Dynamic Grid**

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

- **Grid Major Spacing**

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

- **Sections Per Cell**

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

Units

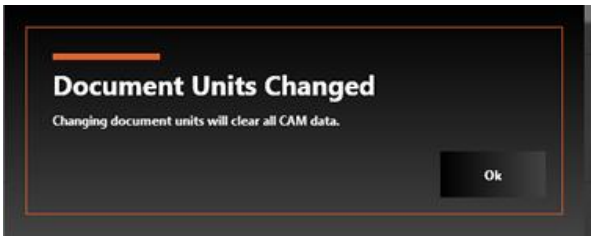
- **Document Units**

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

  - mm – millimeters
  - in – inches

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

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



Cut (Ctrl+X)

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

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

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

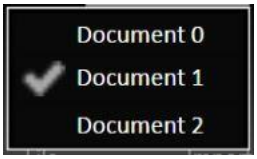
Use the **Select All** command to select all geometry in the workspace.

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.



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.

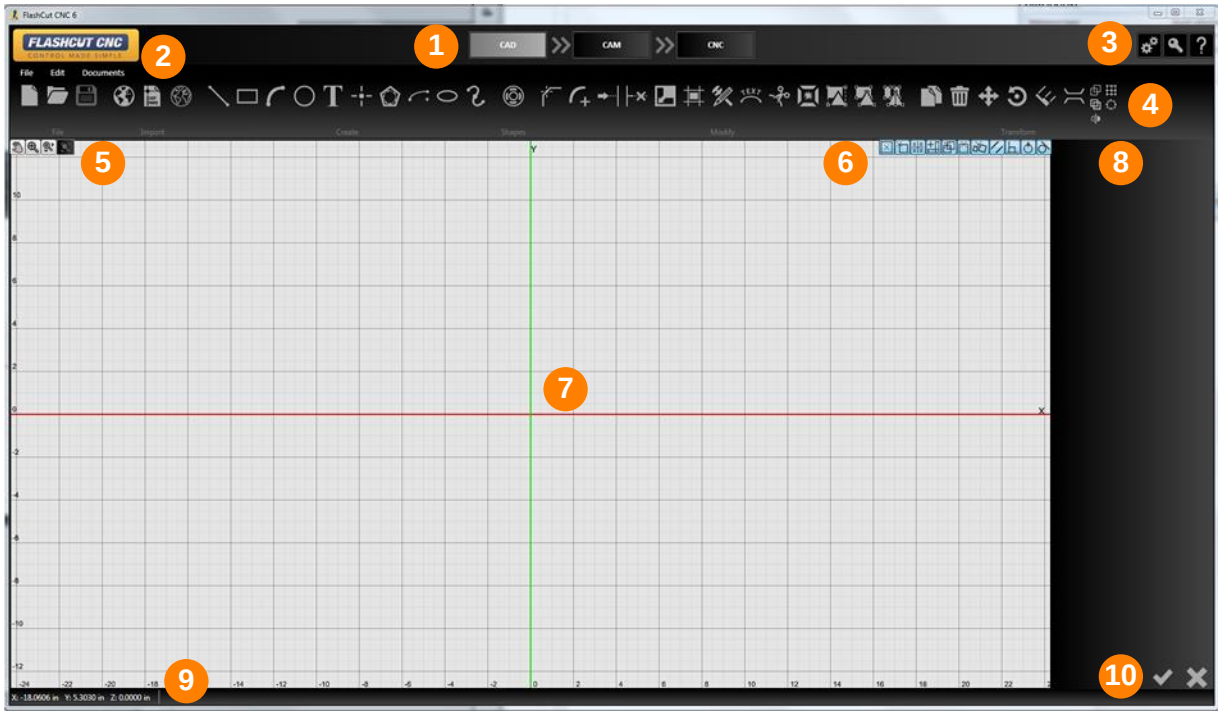
Status bar

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
2. Menu bar
3. Configuration, License, and Help buttons
4. Ribbon
5. Pan and zoom tools
6. Snap tools
7. Drawing workspace
8. Parameters area
9. Status bar
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. 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
- Import tools
- Create tools
- Shape tool
- Modify tools
- 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.

Open (Ctrl+O)

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

Save (Ctrl+S)

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

Import 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 three ways to import a drawing:

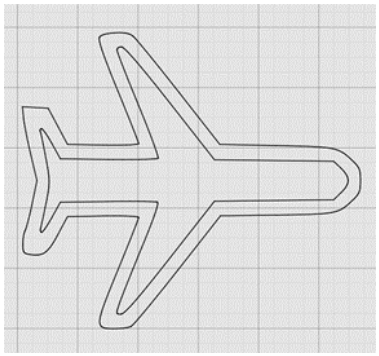
Import silhouette image – creates an outline of the silhouette of an image

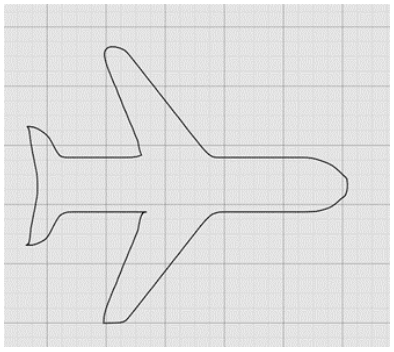
Import DXF/DWG image – imports lines and arcs from a DXF CAD drawing

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

Understanding silhouette and centerline images

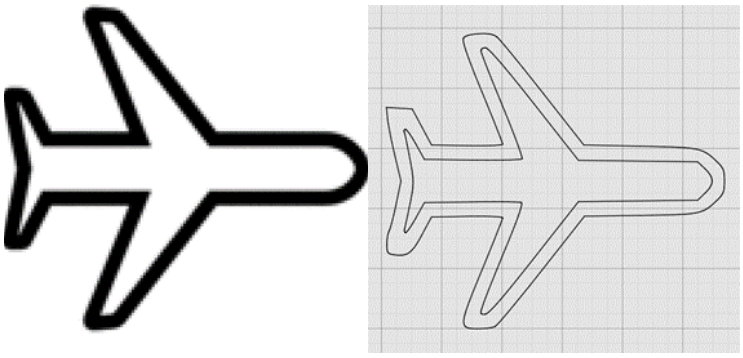
Below are three images: the original line drawing (left), the drawing imported as a silhouette (center), and the same drawing imported as a centerline image (right). When creating a silhouette, FlashCut attempts to identify the drawing by its contrast against the background. Note how both sides of the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



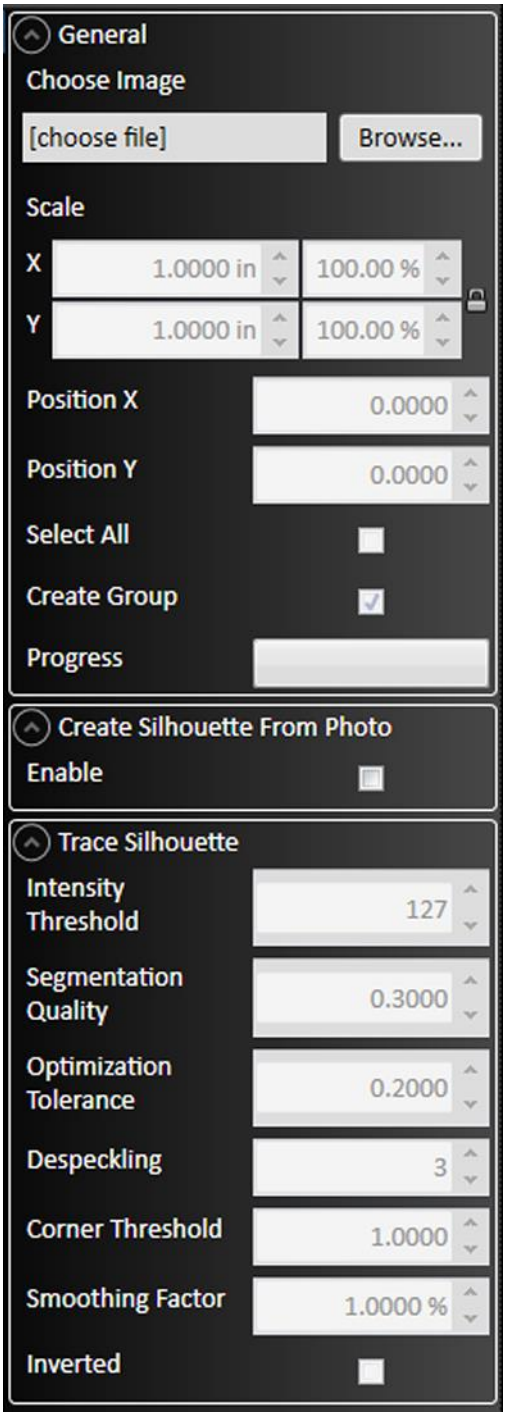


Import silhouette image

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



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



Choose image

Click **Browse...** to select an image file from the computer. Select **Open** to call up the desired image. A list of file formats that can be imported appears on page 24.

Scale

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

Position X

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

Position Y

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

Select All

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

Create Group

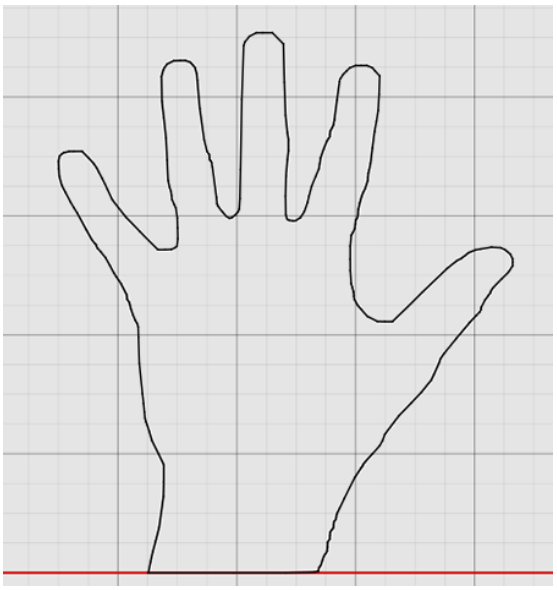
The **Create Group** checkbox allows the user to import the image as a Group instead of importing the image as individual elements.

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.
- **Grid size** – Is the background search resolution. The smaller the grid size the more thorough the analysis of the image, however this requires more computational power and time. Range: 0-10000

- **Color Tolerance** – This tolerance is used to determine the contrast between the background and the silhouette. Range: 0-100.

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: 1-255.

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: 0.0001-100.

Optimization tolerance

Select the degree of accuracy of the lines in the silhouette to the original image. A larger tolerance level allows for more variation from the original drawing, while a smaller tolerance replicates the image very closely. Range: 0.0001-100.

Despeckling

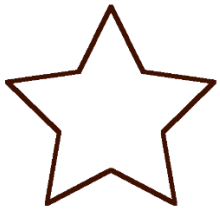
Reduces the number of small dots that appear in the image. It also can reduce the overall detail of the image being imported. The larger the number, the lower the dot tolerance, causing fewer dots to appear in the imported silhouette. Range: 0-100.

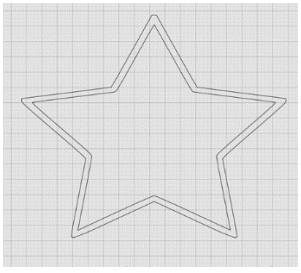
Corner threshold

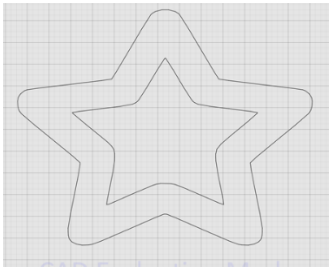
Select the tendency of FlashCut CAD to generate rounded corners at line vertices. Increasing this parameter will make the corners less jagged, but sharper features will become rounded. Range: 0.0001-100.

Smoothing Factor

Reduces the magnitude of small jagged features on an entity. It also can reduce the overall detail of the image being imported. The larger the factor, the smoother the result. The examples below show the effects of the minimum and maximum amount on the original (left). Range: 0-5%.



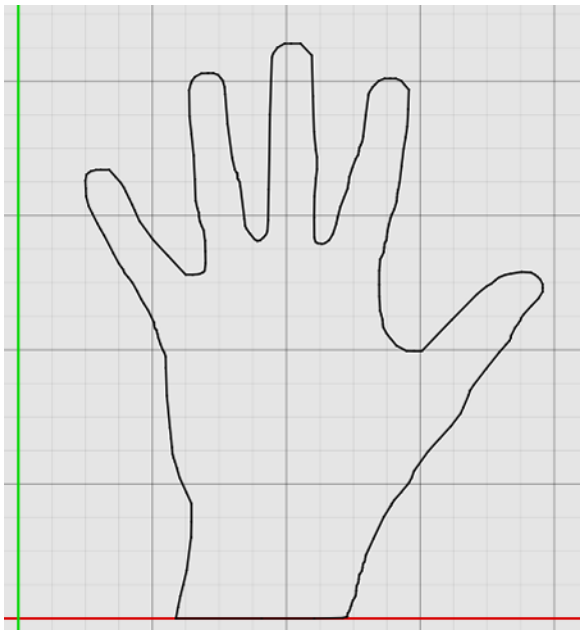


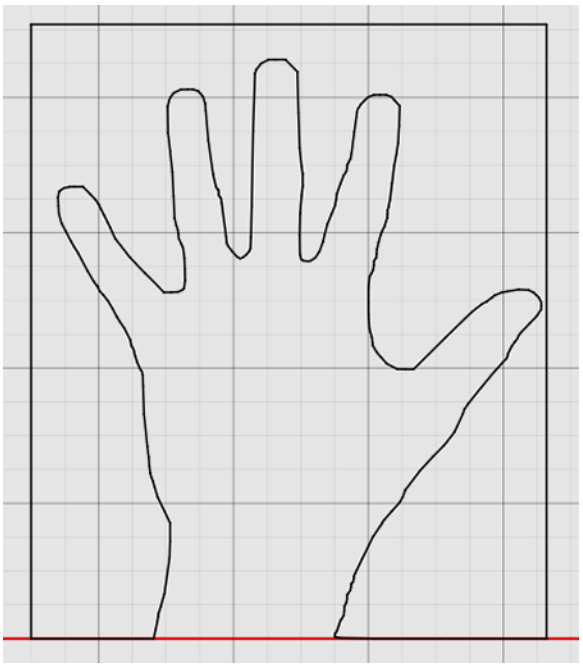


Inverted

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

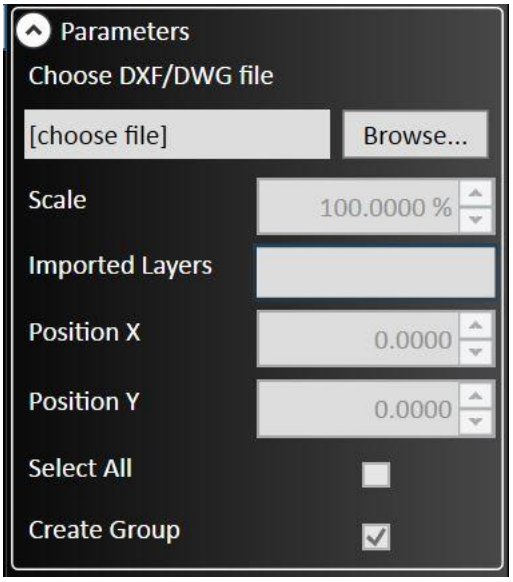
For example, both objects below were imported as silhouettes from a photo. The right hand object is an inverted version of the left. The left hand object is a part; the right hand image is a cutout.





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.

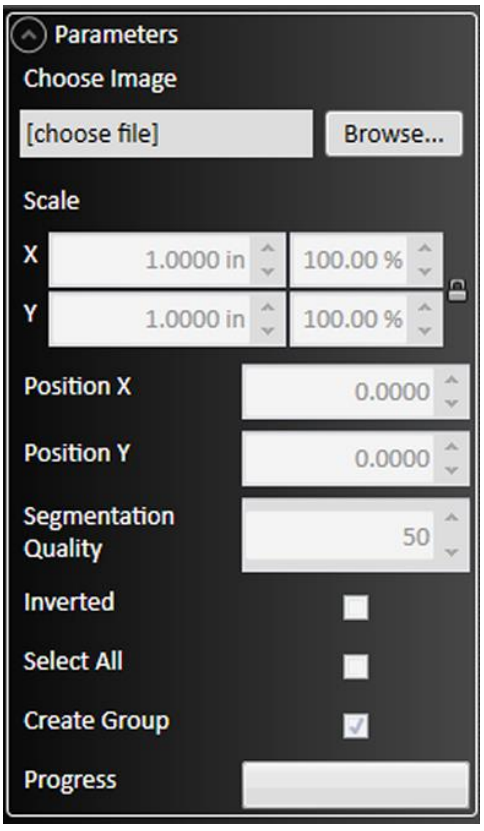
Create Group

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

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 24.

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

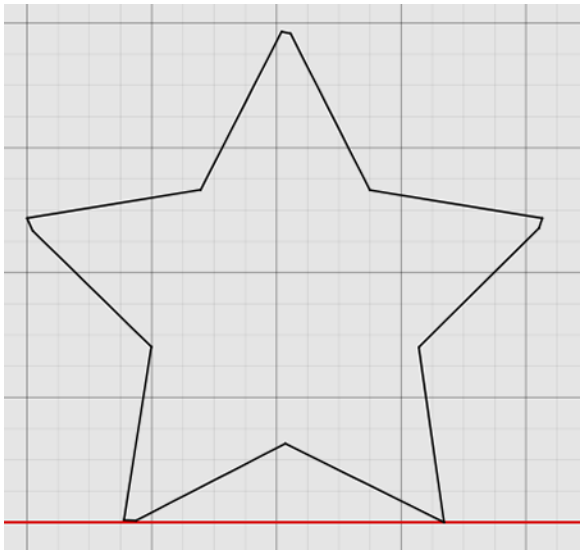
Enter the value for the Y coordinate of the lower left corner of the imported image.

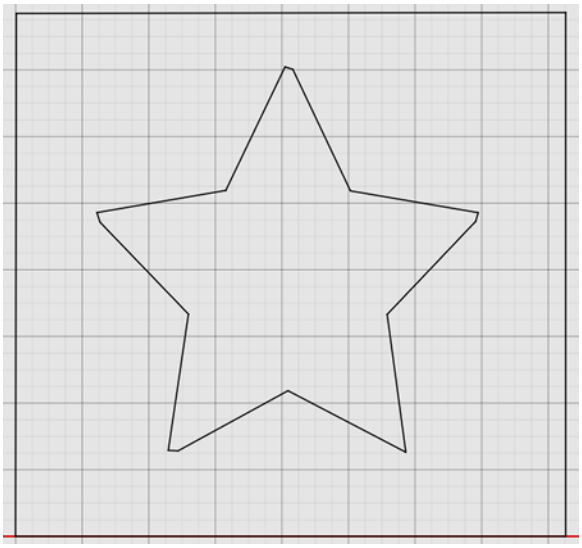
Segmentation quality

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: 0.0001-100.

Inverted

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





Select all

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

Create Group

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

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 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.

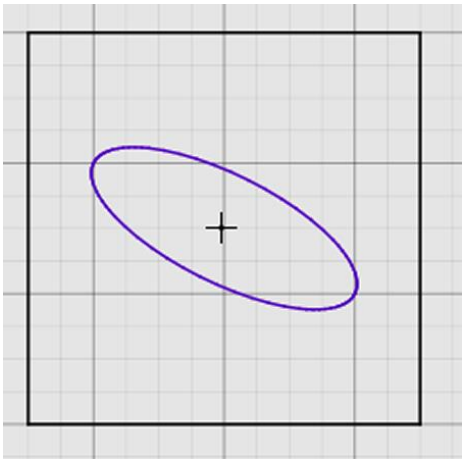
Each tool (with the exception of Point) lets you define the **Treatment** parameter to determine whether the element is cut or only marked.

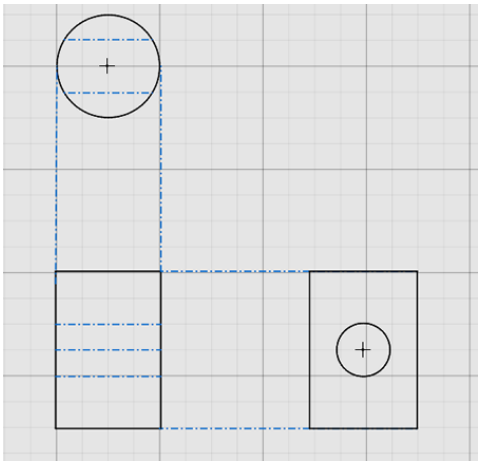
Select **Cut Element** to use the default cutting fab head. This is the pre-selected option. These elements appear in the CAD window as solid black lines.

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.

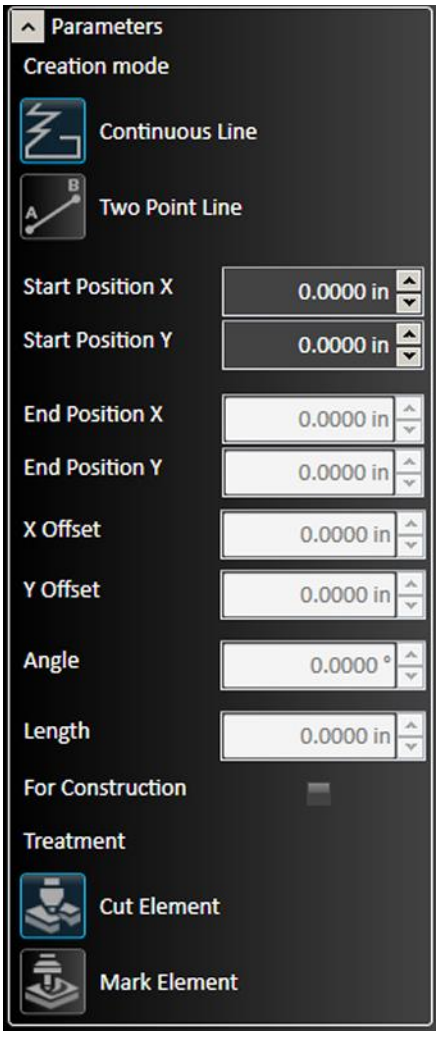
These are the tools that are available on the **Create** portion of the ribbon:

- **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.

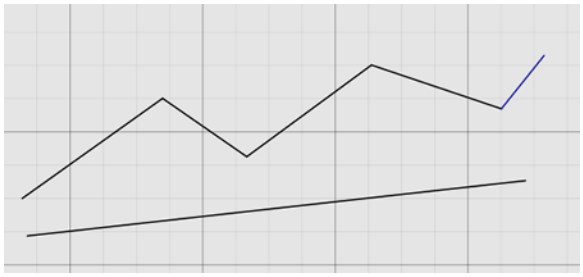


Creation mode

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.

For Construction

Select to make the line a construction line.

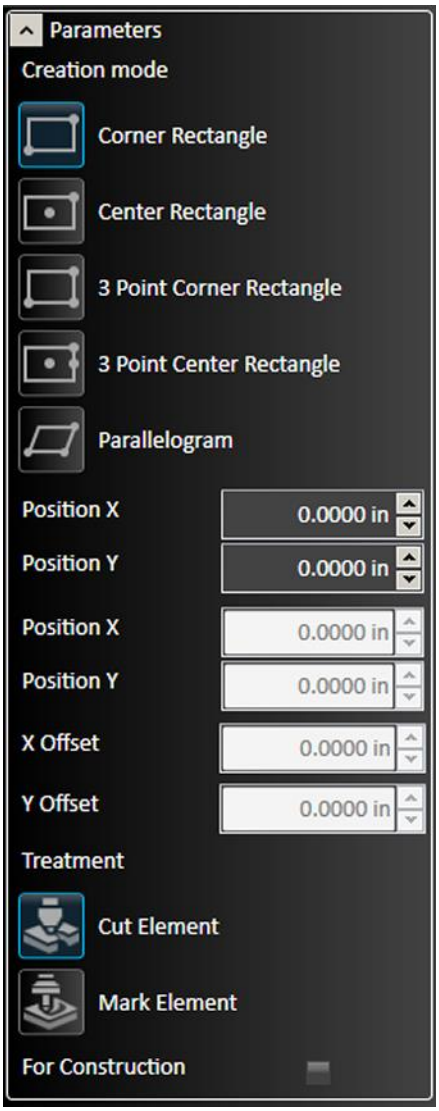
Treatment

Select whether the element should be cut or marked.

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.

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

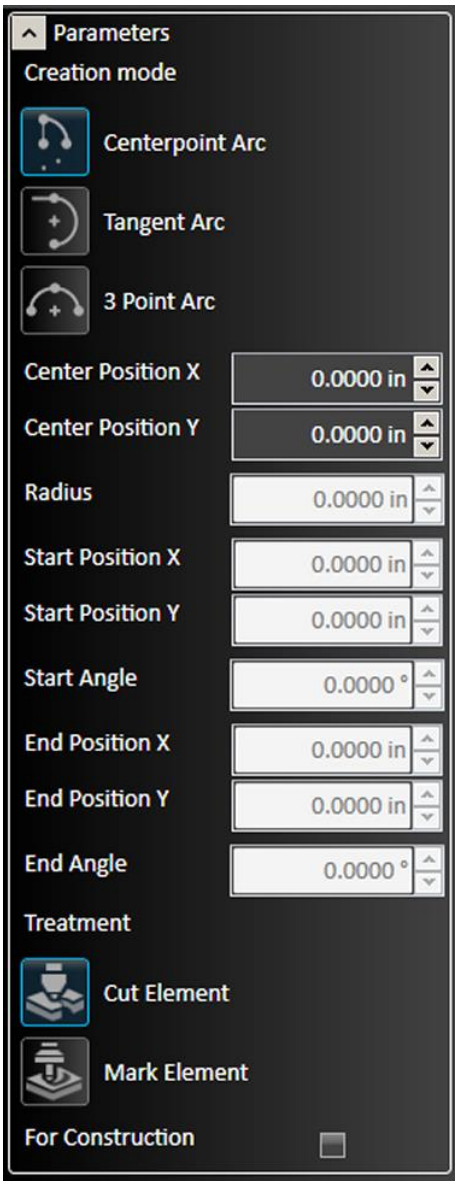
Specify the attributes of the arc.

Treatment

Select whether the element should be cut or marked.

For Construction

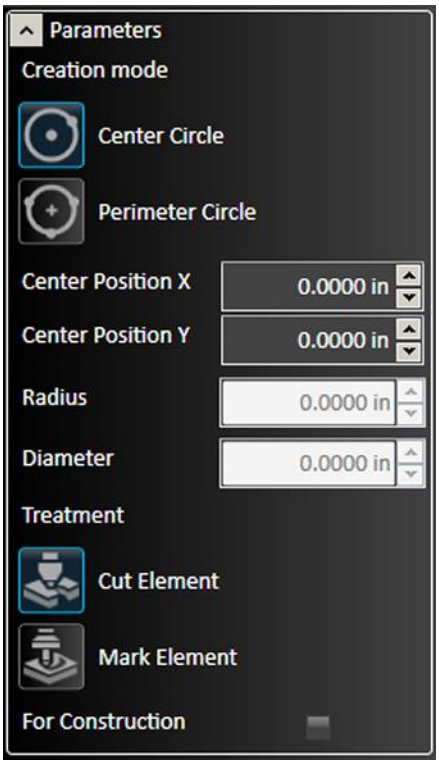
Select to make the arc a construction line.



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.

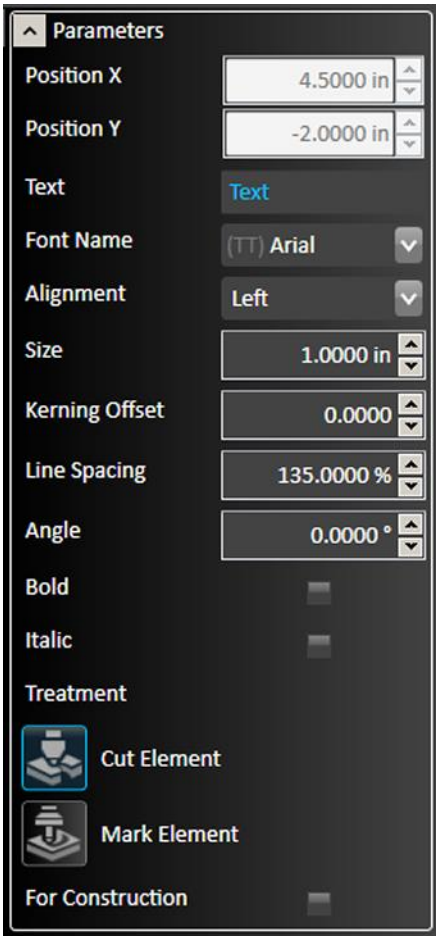
For Construction

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.

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.

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.

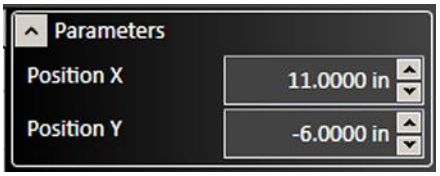
For Construction

Select to make the text object a construction line. You may also select options for bold and italic here. 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.

Point

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

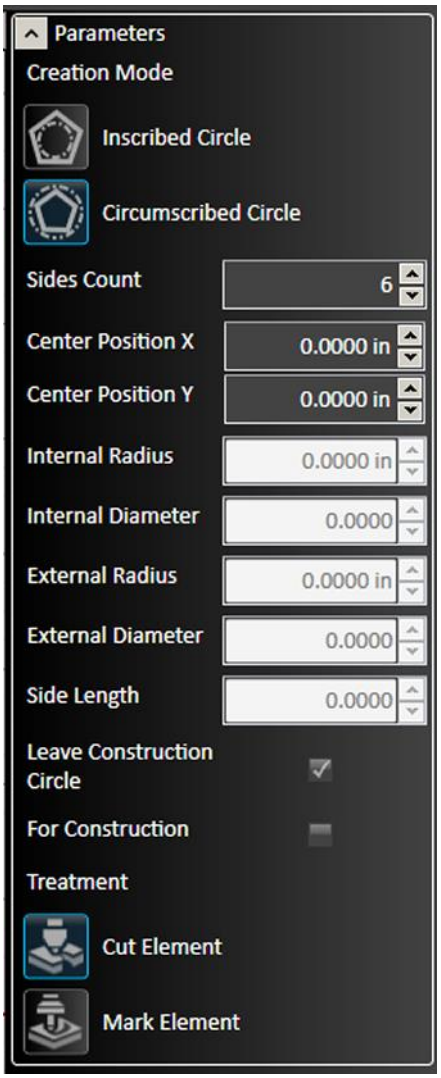


Position X and Position Y

Specify the position of the point relative to the origin.

Polygon

FlashCut CNC enables you to select the number of vertices setting, and whether the shape will be inscribed or circumscribed within a construction circle.



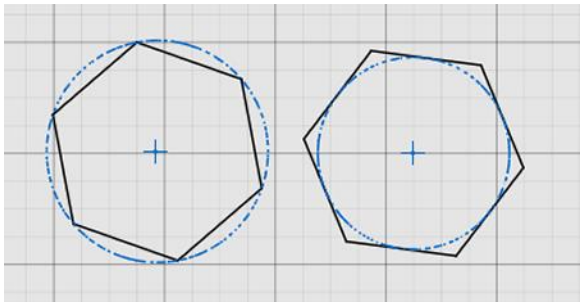
Creation mode

Select the type of polygon to create.

- **Inscribed Circle**
- **Circumscribed Circle**

Click in the drawing window to select the center point of the polygon. Drag out to the point that will be the first vertex of the polygon.

Examples of each type are shown below.



Sides Count

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

Position, radius, diameter and side length

Specify the attributes of the polygon.

Leave Construction Circle

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

For Construction option

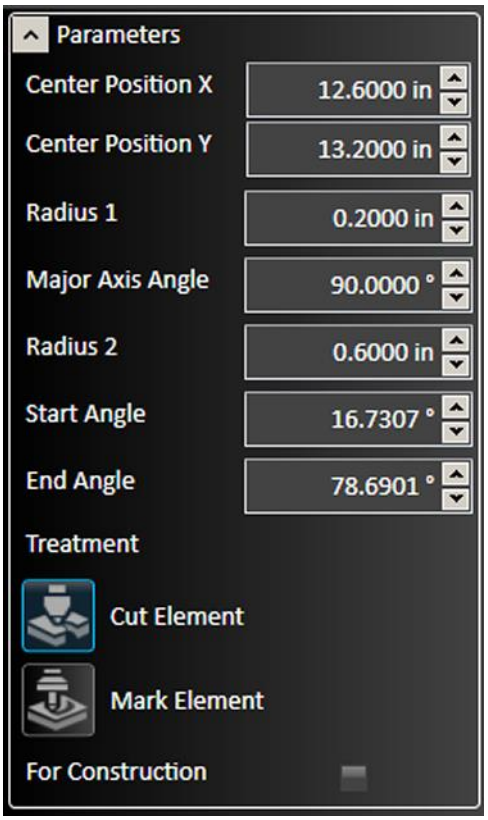
Select to make the polygon a construction line.

Treatment

Select whether the element should be cut or marked.

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.

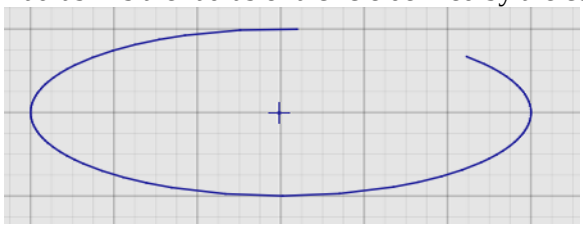
Finally, select a start point and then an end point to the arc.

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

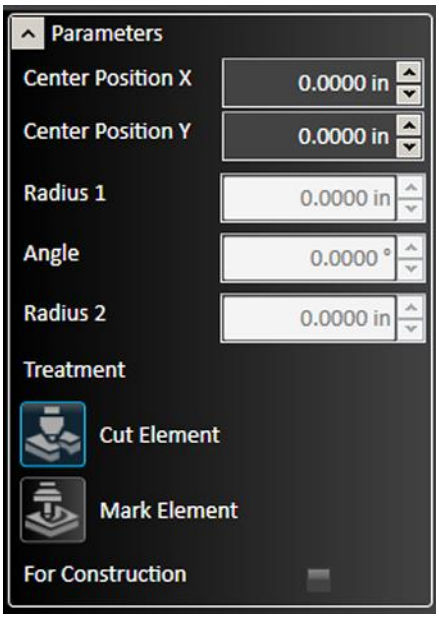
Select whether the element should be cut or marked.

For Construction option

Select to make the line a construction line.

Ellipse

Creates an ellipse.



Drawing the ellipse

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

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

Center Position X and Center Position Y

Specify the coordinates of each center point.

Radius 1 and Radius 2

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

Angle

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

Treatment

Select whether the element should be cut or marked.

For Construction

Select to make the ellipse a construction line.

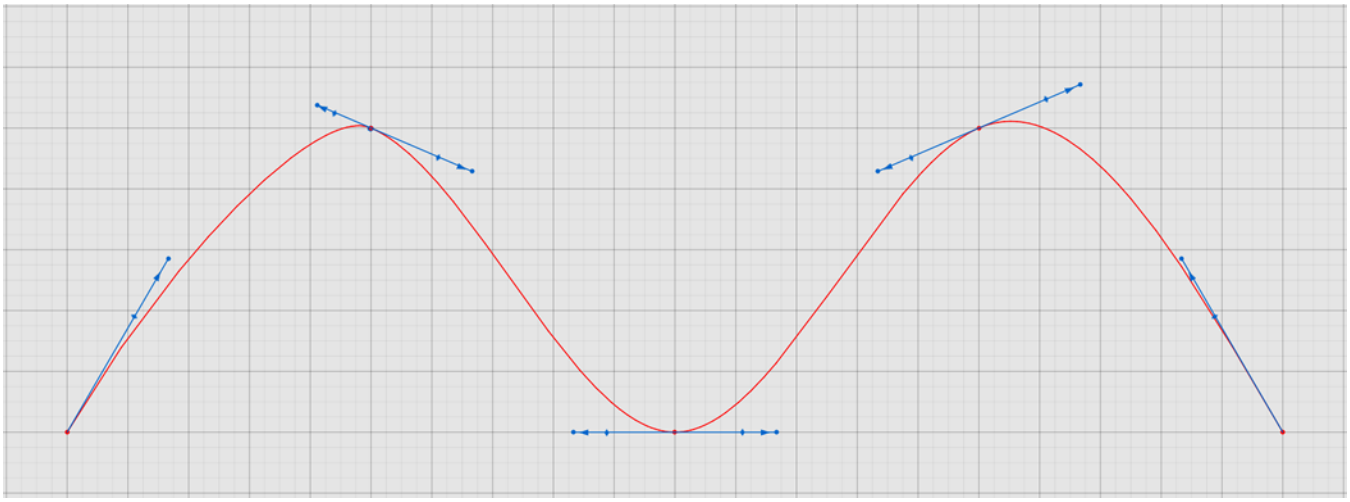
Spline

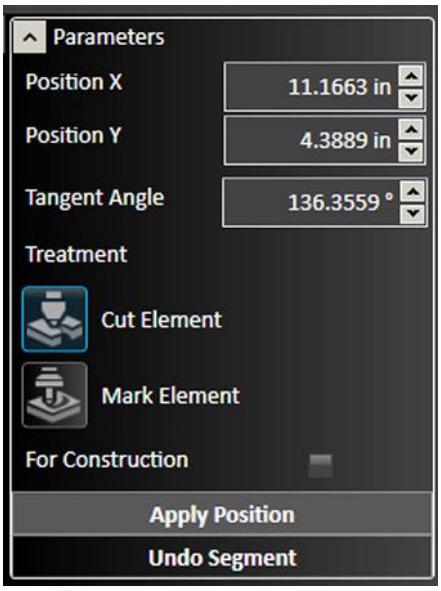
Creates a complex curve in multiple segments.

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

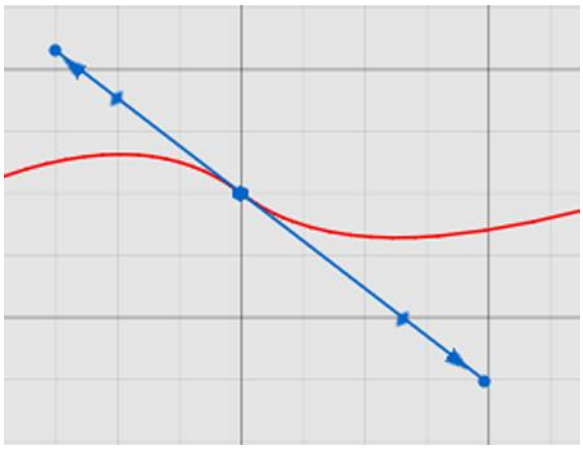
The default function of each click: the new line segment is added.

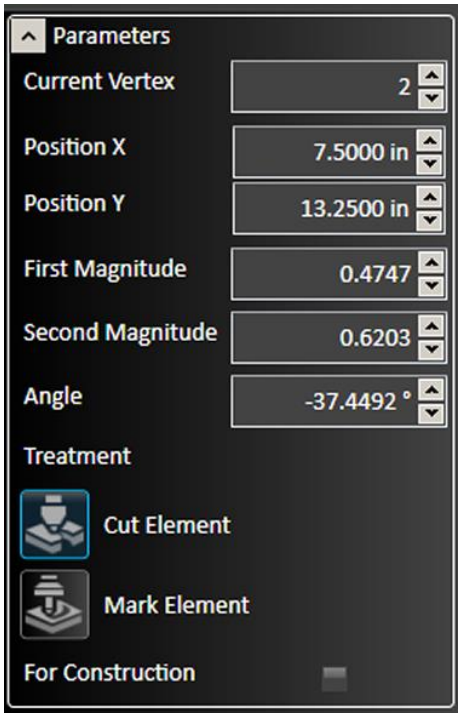
Undo Segment

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

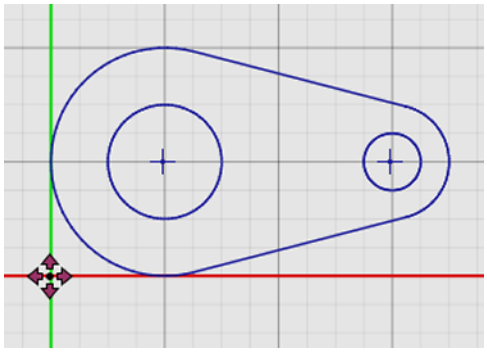
Select to make the line a construction line.

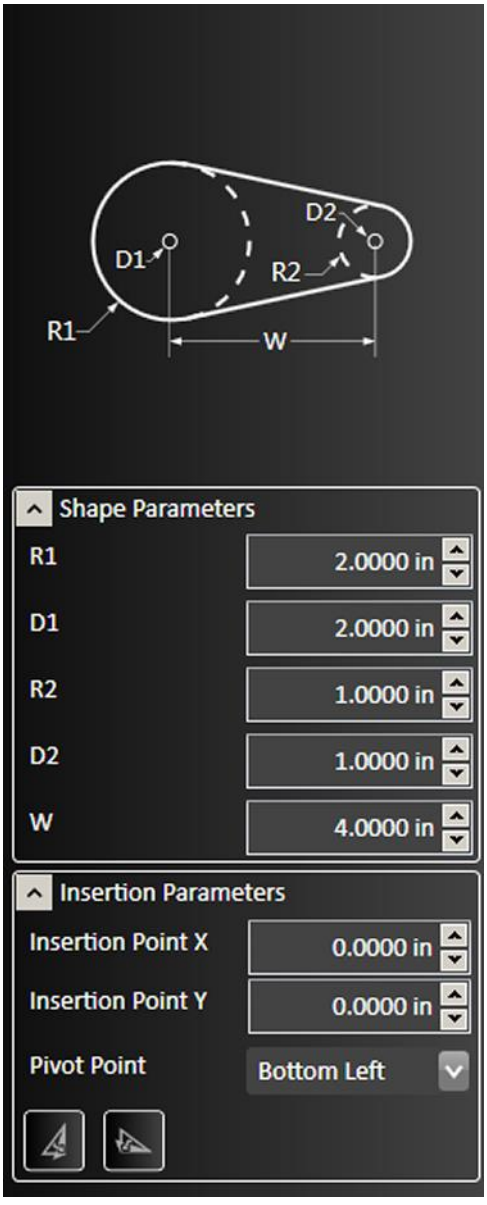
Shape tool

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).

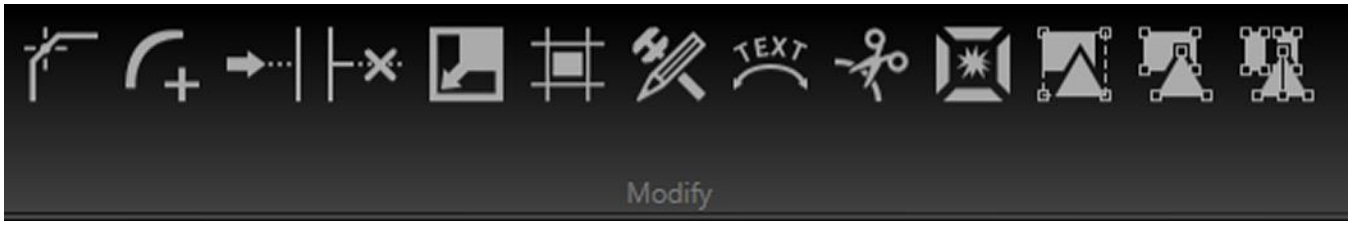
Insertion parameters

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.

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.

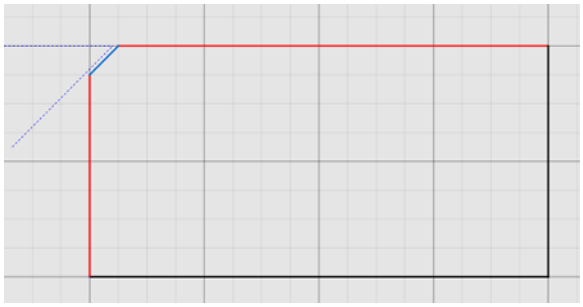
These are the tools that are available on the **Modify** portion of the ribbon:

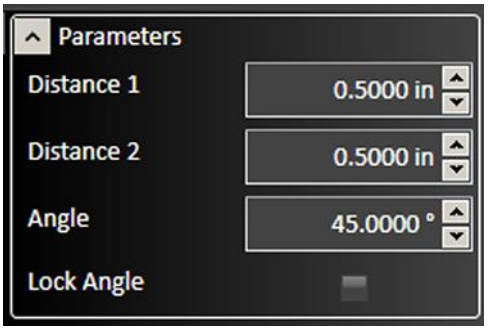
- Apply Chamfer
- Apply Fillet
- Extend Tool
- Trim Tool
- Scale Entities
- Apply Offset
- Fix Drawing
- Curve Text
- Cut Tool
- Explode Entities
- Group
- Ungroup
- Ungroup All

Apply Chamfer

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

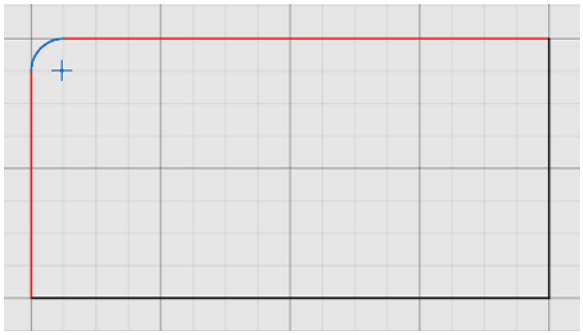
Choosing the **Lock Angle** option causes both distances to change when one of them is edited, holding the angle measurement constant. Once the chamfer is correctly configured choose the green check mark or press ↵**Enter**.



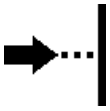


Apply Fillet

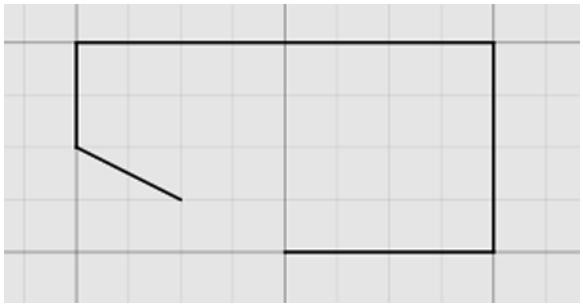
Select two intersecting lines or their intersection point, and then click **Apply Fillet** to add a rounded fillet to this edge. You can scale the size of the fillet by dragging the mouse towards or away from the fillet, or in the parameter window, type in the radius of the fillet. Once the fillet is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

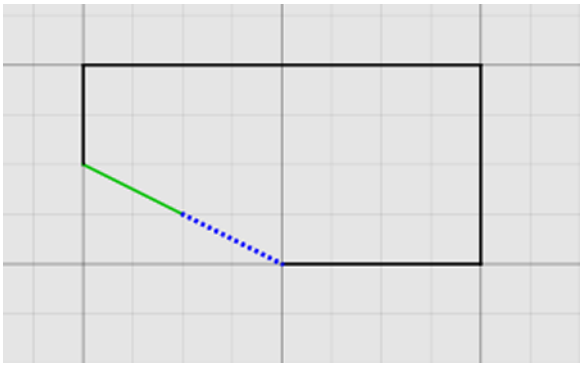


Extend Tool



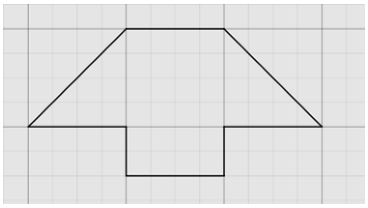
Click the **Extend Tool**, and then hover the selection cursor over the endpoint of the line or curve to be extended. A preview of the extension should appear (in blue, below) to show where the line or curve will intersect with another object in the drawing. Click the mouse to finalize the extension.

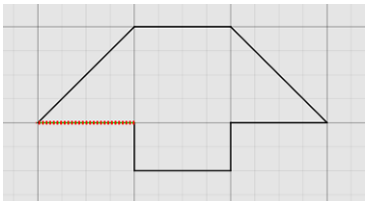




Trim Tool

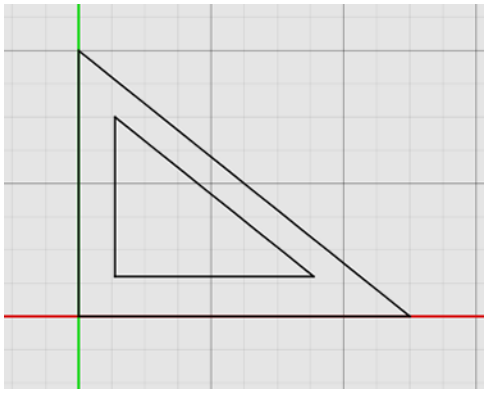
Click the **Trim Tool**, and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear (in red, below) where the line or curve will be removed. Click the mouse to finalize the trim.

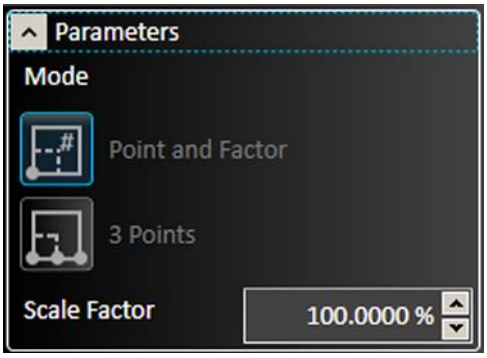


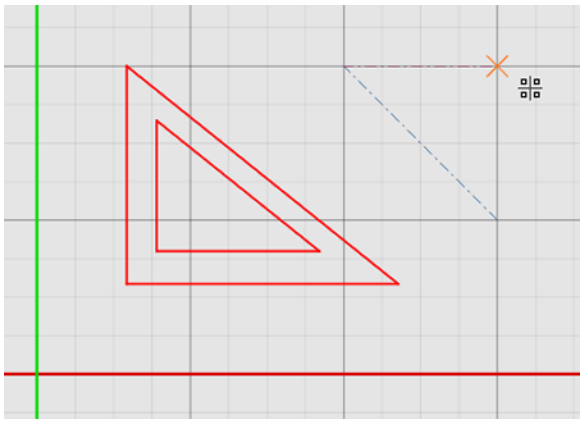


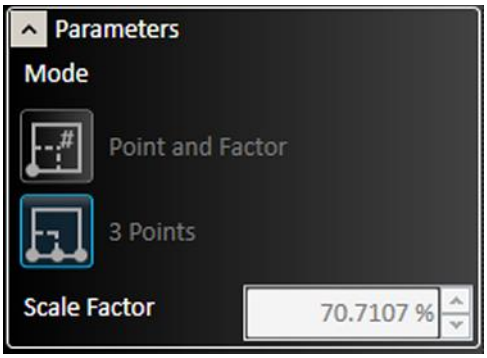
Scale Entities

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.

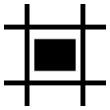






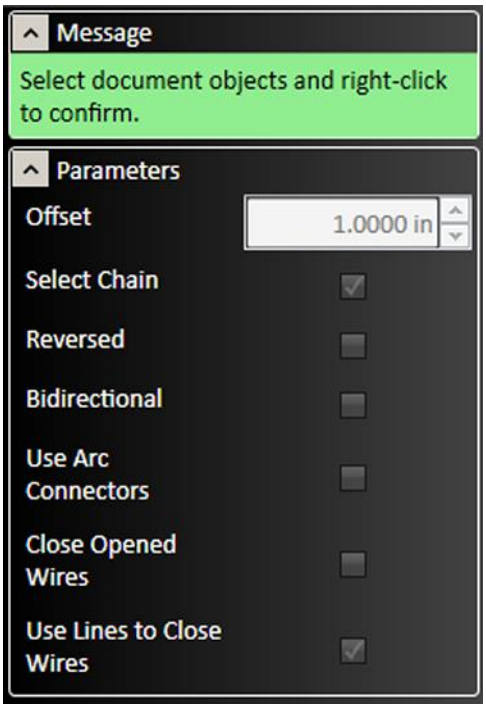


Apply Offset



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

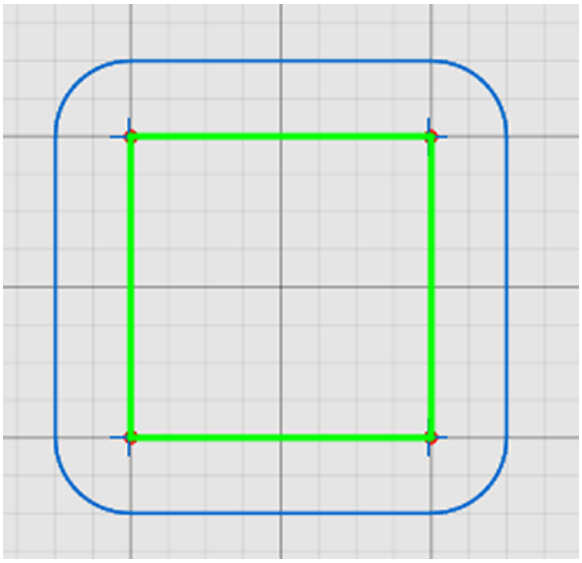
Bidirectional creates a second copy, opposite the first. When the **Bidirectional** option is chosen, the **Close Opened Wires** option appears.

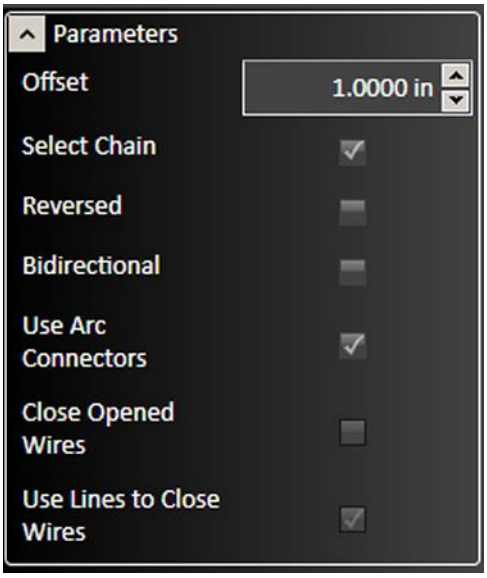
Close Opened Wires

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.





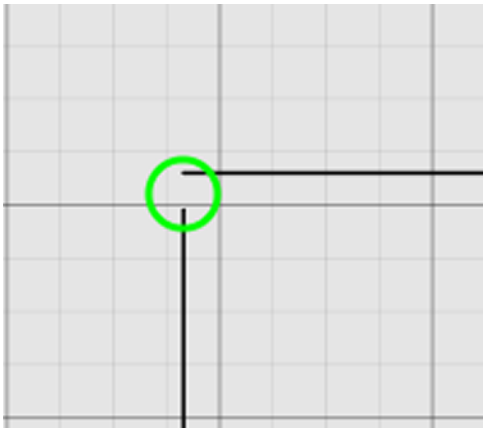
Use Lines to Close Wires

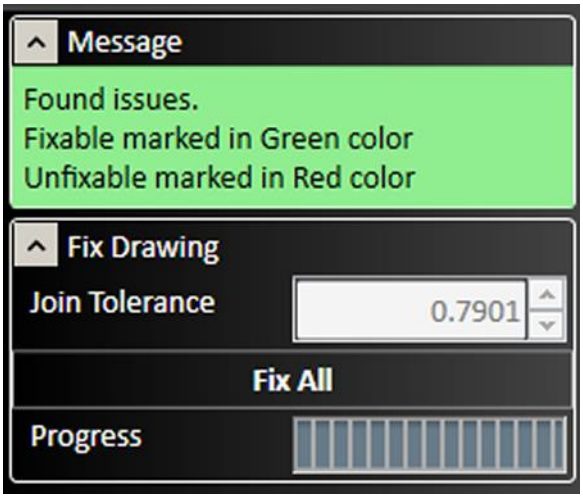
Use Lines to Close Wires changes the rounded edges to flat, straight lines.

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** larger than any of the errors in the drawing.





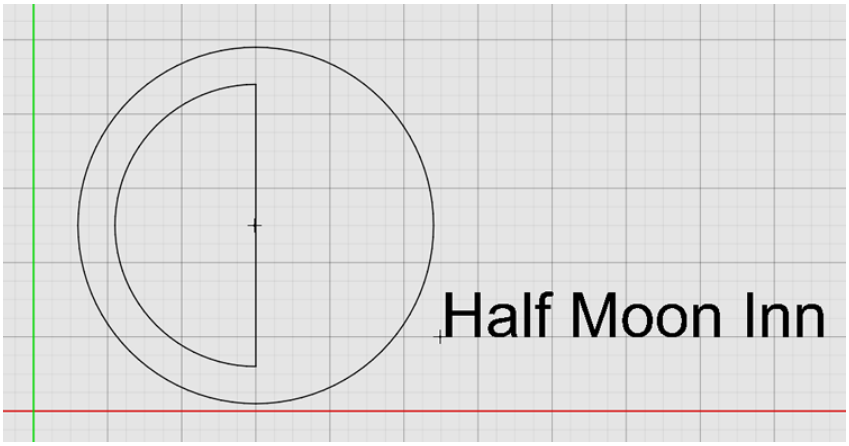
Use the **Search Issues** button to identify issues with the drawing that could lead to mistakes in fabrication. Issues noted with a green circle can be fixed automatically by selecting **Fix All**. Issues noted with a red circle must be addressed manually.

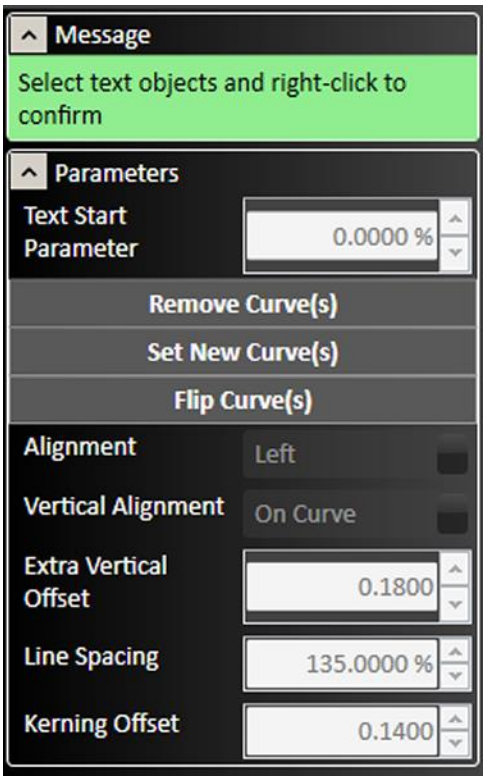
Curve Text

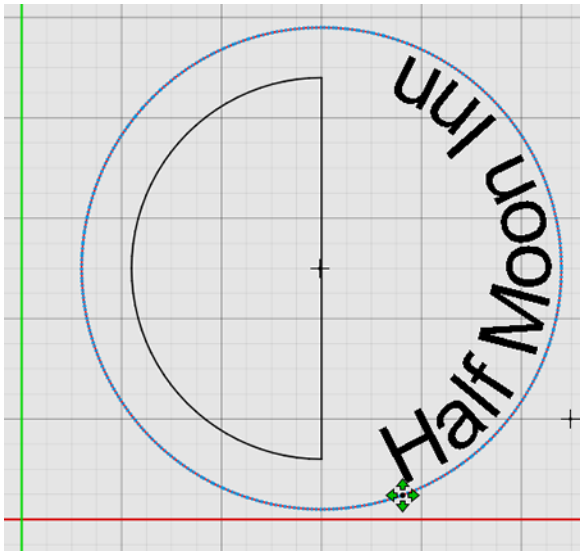
The **Curve Text** tool modifies an existing text object to follow a curved shape. Start by creating both the text object and the curved shape you want to use.

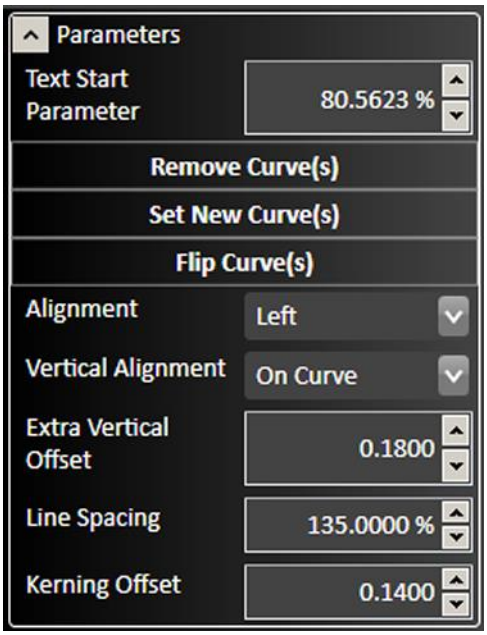
Select the Curve Text tool and follow the instructions in the Parameters window. Select the curve, and then right click on the text object. The position of the text object can be adjusted with the cursor, and modified by changing the parameters.

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.

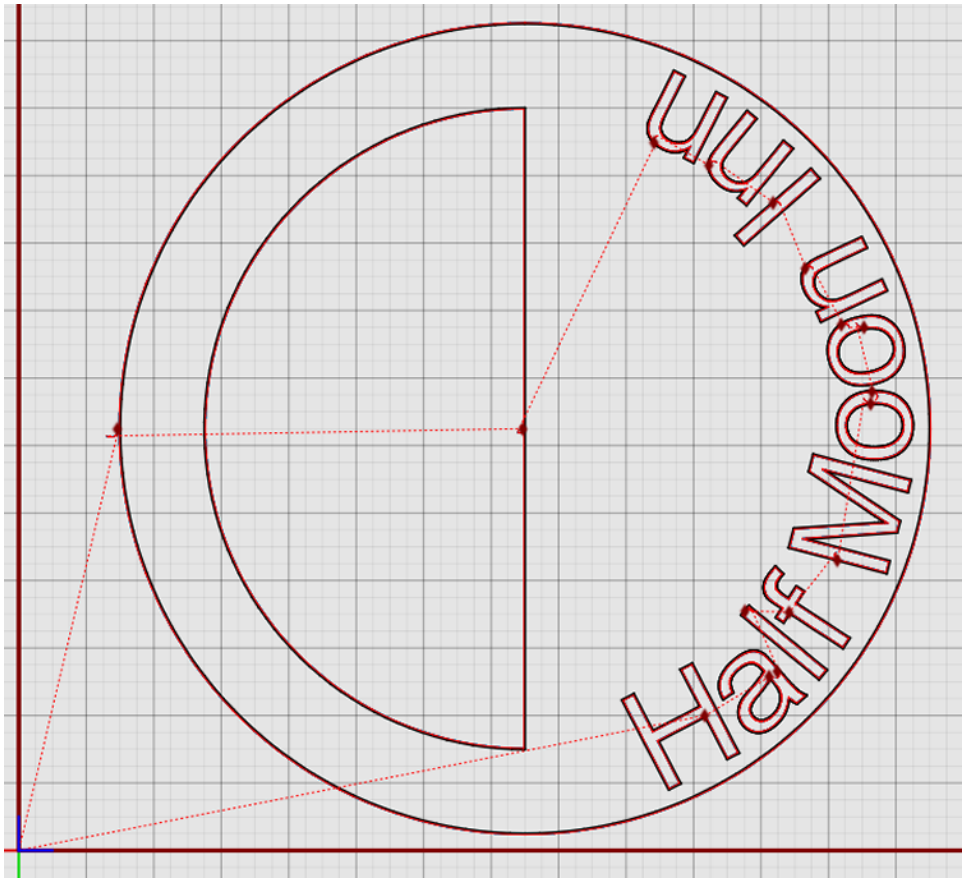








The resulting CAM toolpath for the sample drawing appears below. Note that it is not necessary to explode letter shapes to create toolpaths.



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)

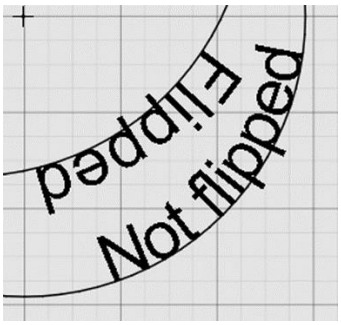
Separates the text object from the curve.

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.



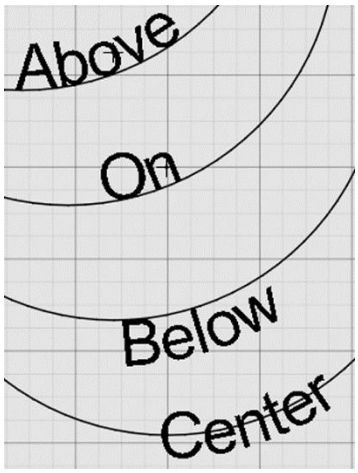
Alignment

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

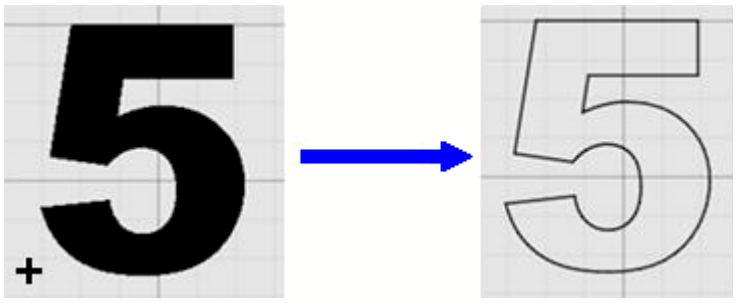
Use the **Cut Tool** to divide an entity such as an arc or a line segment into multiple sections.

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.

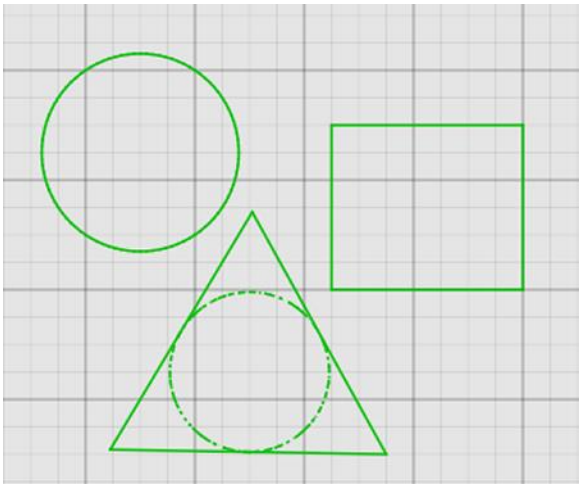
If you choose this command without a group selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking. See

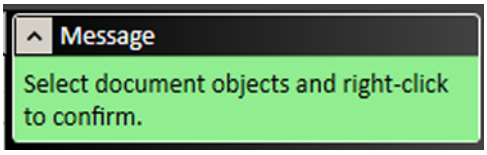
Ungroup All

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.

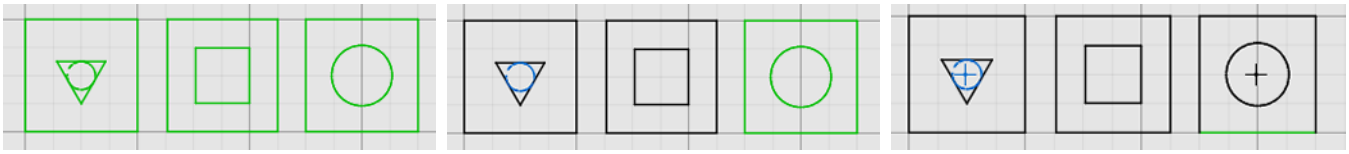
The example below shows the highlighting and message for all grouping operations.





Ungroup vs. Ungroup All

The difference in behavior between Ungroup and Ungroup All is shown below. Each object in the group (left) is itself a group of multiple objects. **Ungroup** (center) will separate the group but leave each subgroup intact, while **Ungroup All** (right) will break each object into its component lines. Note how the selections (green) indicate the level of grouping: all, some, none.



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
- 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

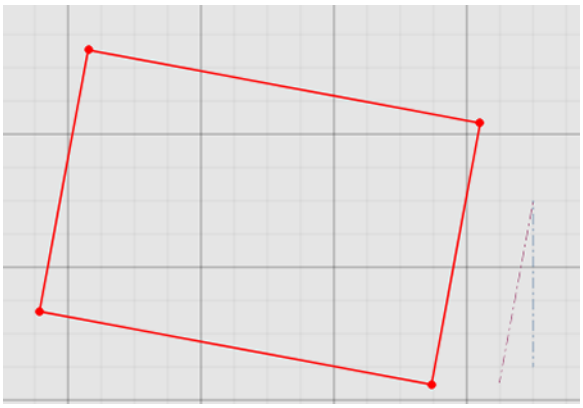
Select the features to be moved, and then click the **Move Entities** tool. Click the mouse to set a reference point, and then move the cursor to move the feature(s) in relation to that point. Click the mouse again to select the new location for the features.

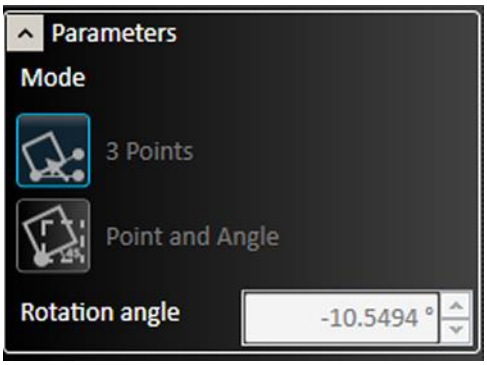
Rotate Entities

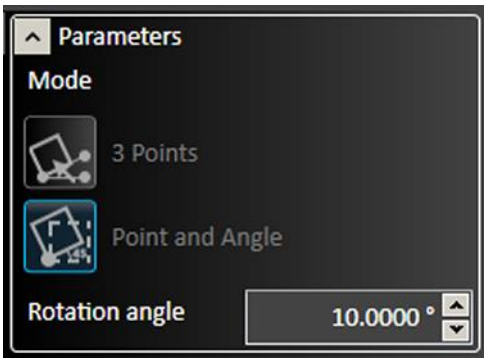
FlashCut CNC offers two modes for rotating features.

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

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

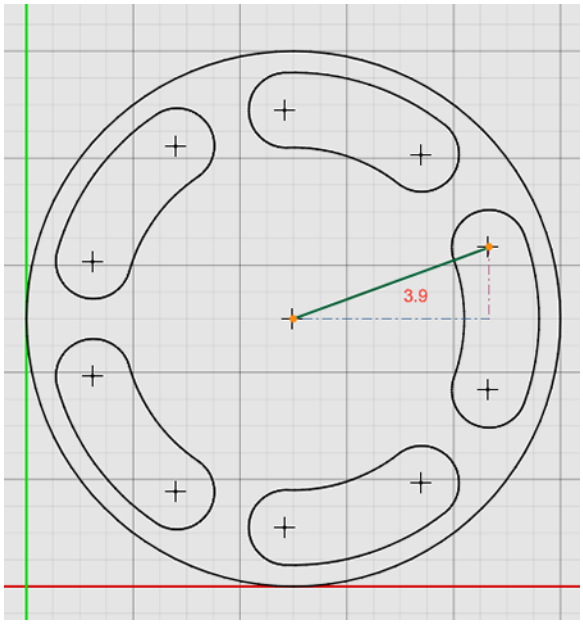


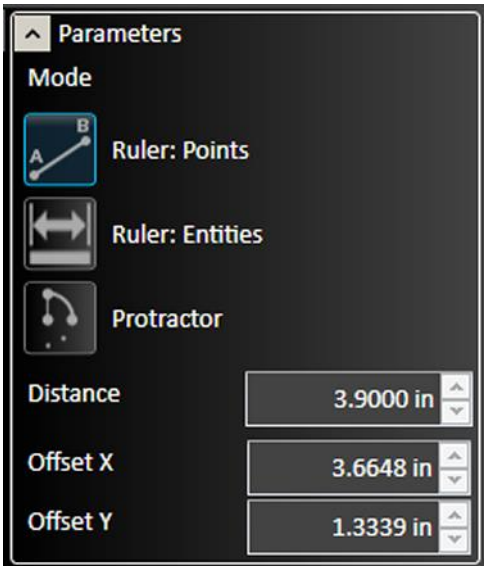




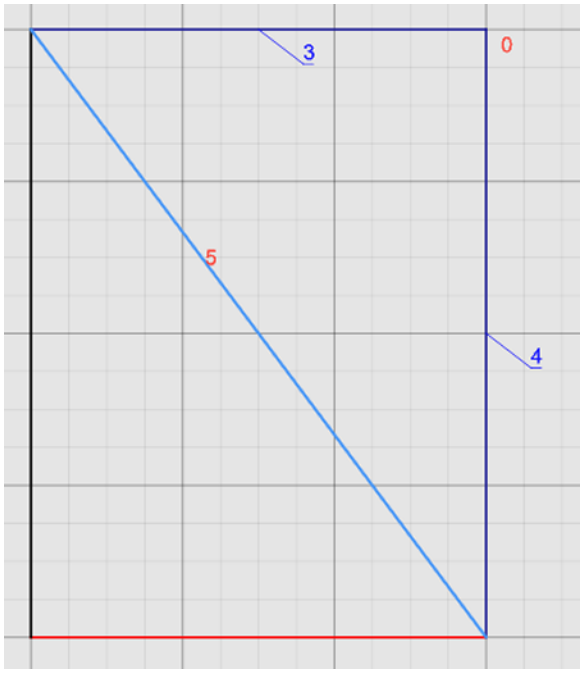
Measure Entities with Ruler

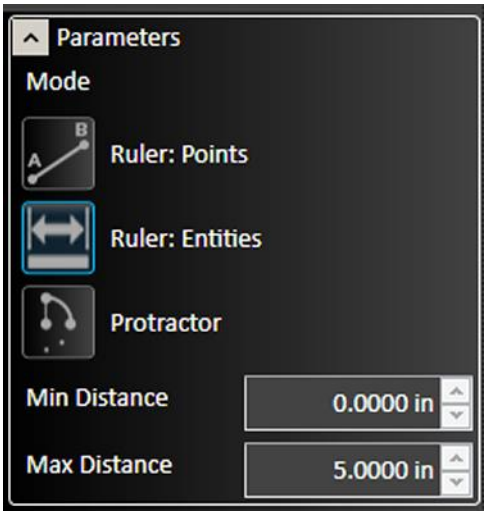
You can measure entities in one of three modes: as a **ruler** between **points** or between **entities**, or as a **protractor** between points on an arc. The tool allows for measurement between any two points, and displays offset in X and in Y directions.



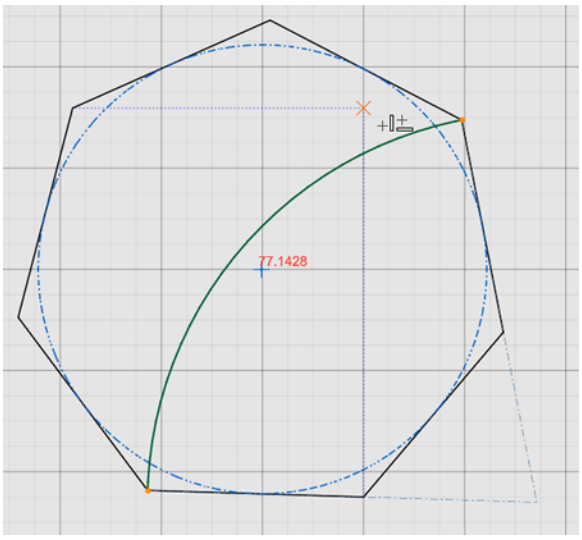


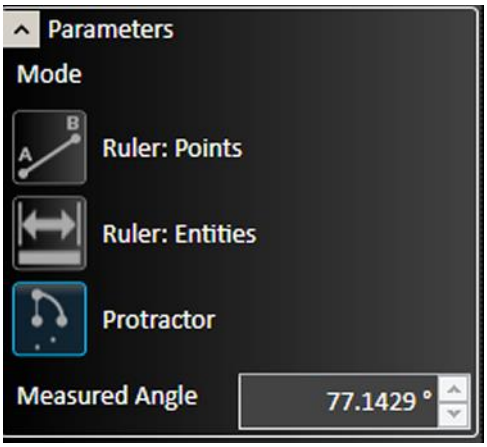
Ruler used to measure between the distance between points: center of an arc, and radius of an arc. Note the construction lines marking the other sides of the triangle: these represent the **Offset X** and **Offset Y**





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



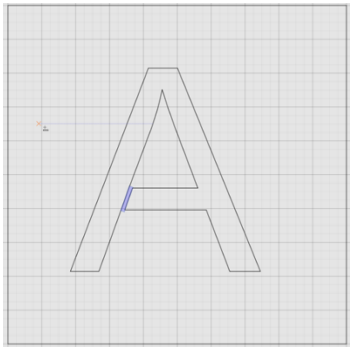


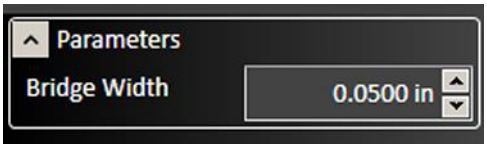
Protractor used to measure the arc and **Measured Angle** between two points of a polygon. Note that the protractor will not measure an arc or angle if it is grouped with other lines or shapes. You must ungroup the arc to enable the protractor tool.

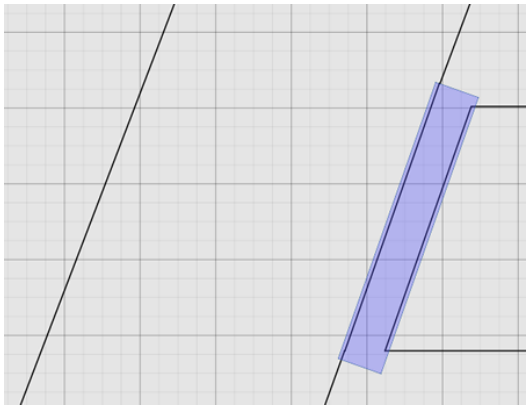
Bridge Entities

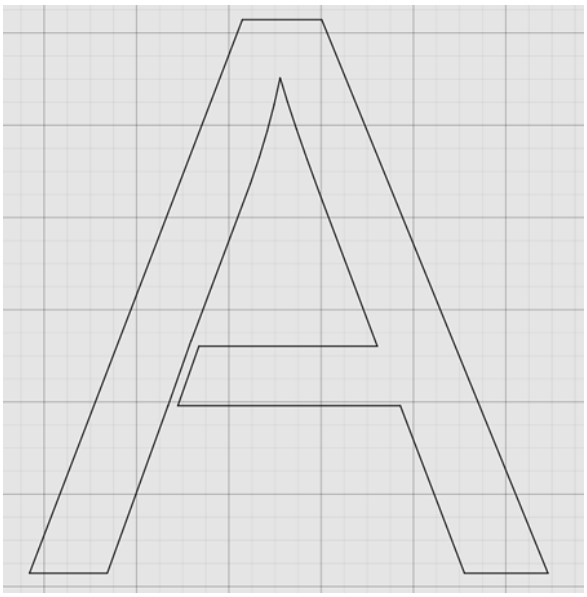
Connects two or more objects with a bridge that spans the gap between them. Can also be used to divide a single object into separate objects. The width of each bridge can be set individually.

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





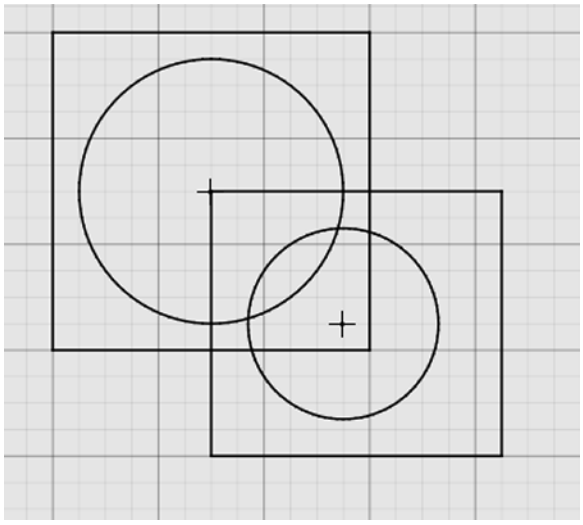


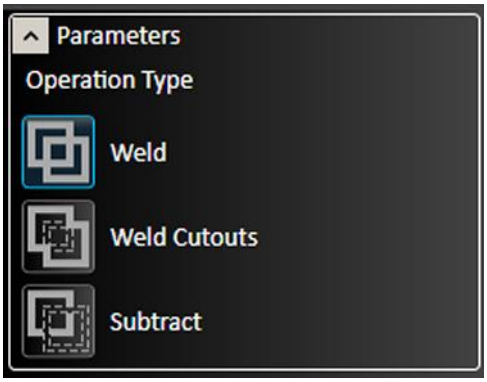


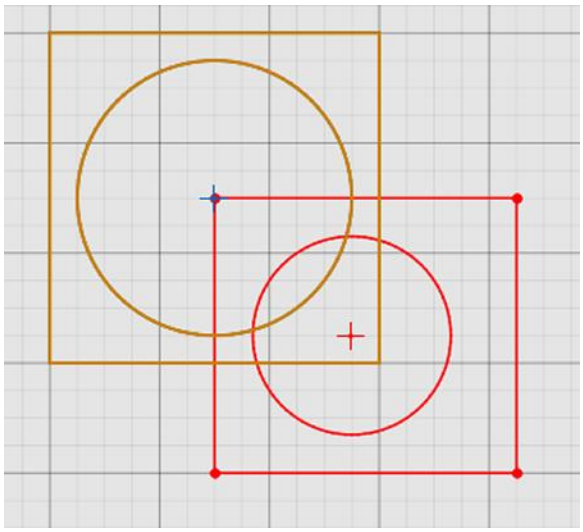
Perform Boolean Operation on Entities

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

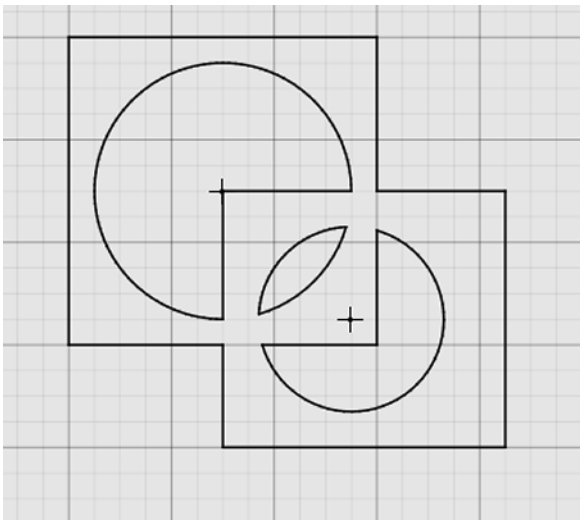
Select Perform Boolean Operation, select the specific operation in the parameters window, then select the objects one at a time, right clicking to confirm each selection. The first object selected will be modified by the second. The example below shows the results of all three operations on the initial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



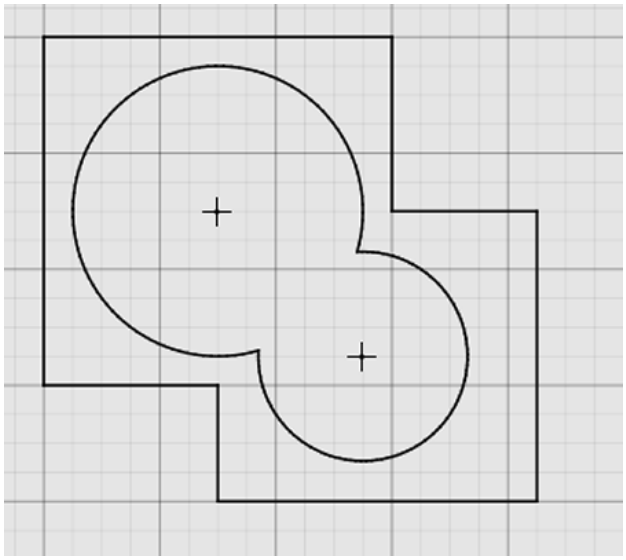




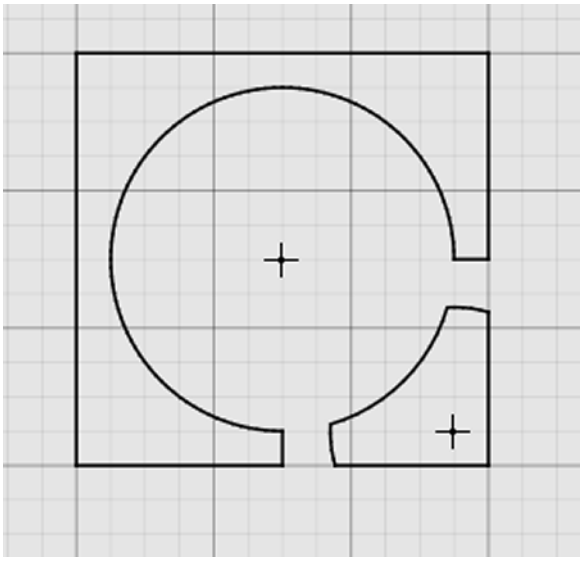
Shapes highlighted to indicate which object is first (gold) and second (red).



The results of a **Weld** operation.



The results of a **Weld Cutouts** operasion

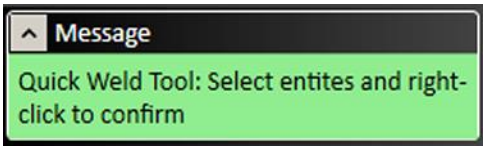


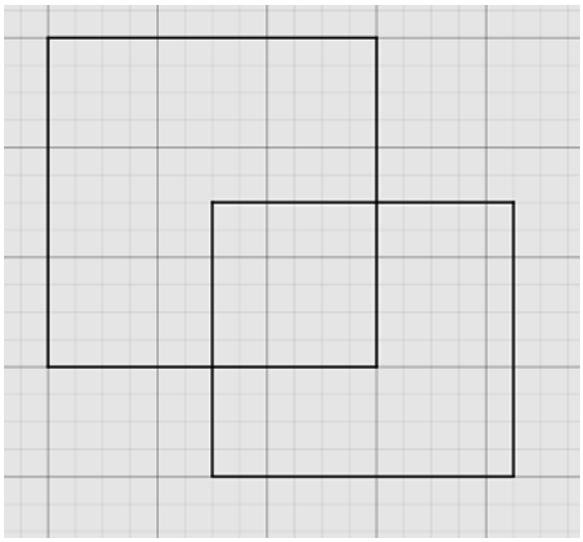
The results of a **Subtract** operation.

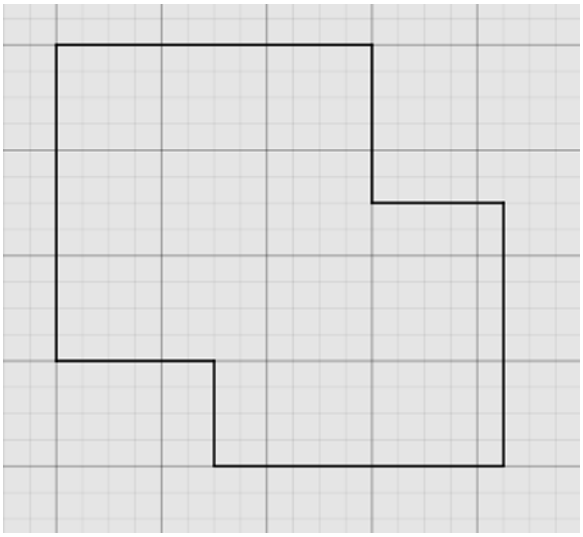
Weld Selected Entities

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

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

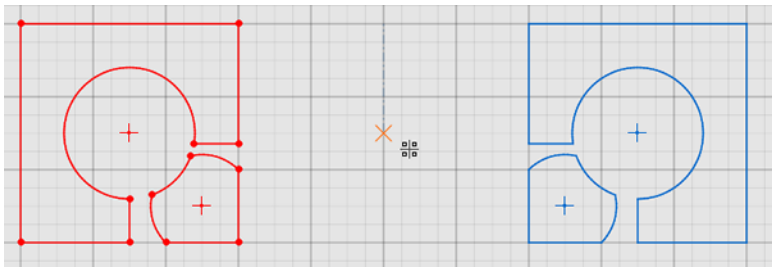


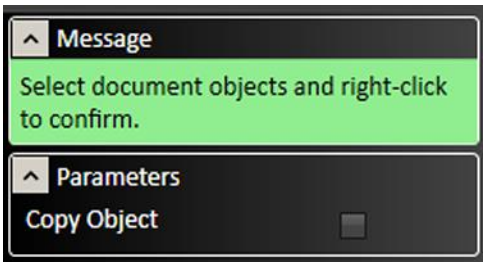




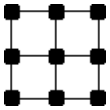
Mirror Entities

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





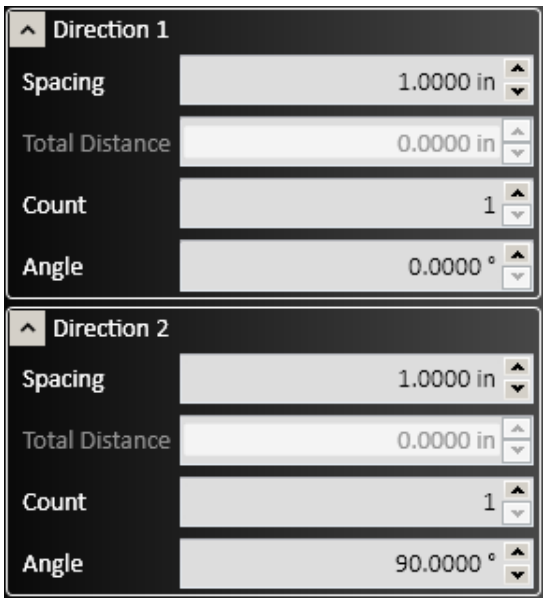
Linear Pattern Tool

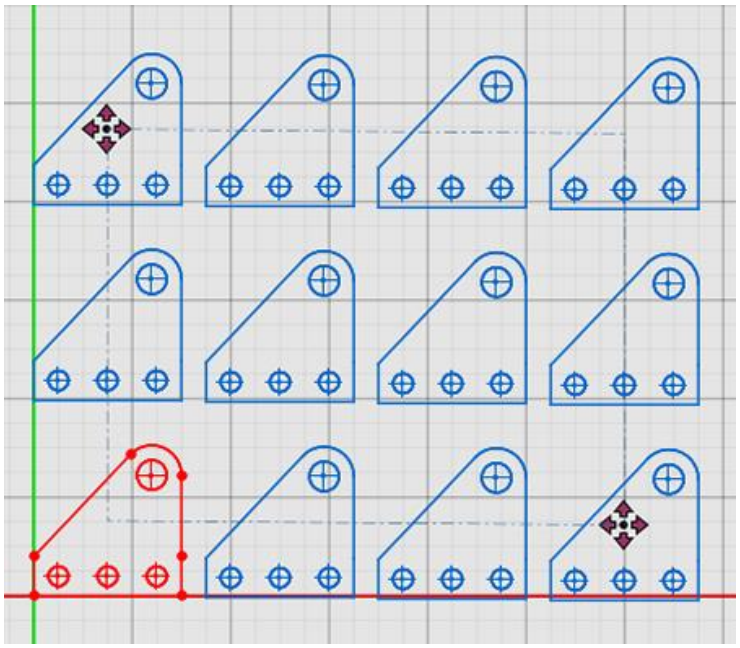


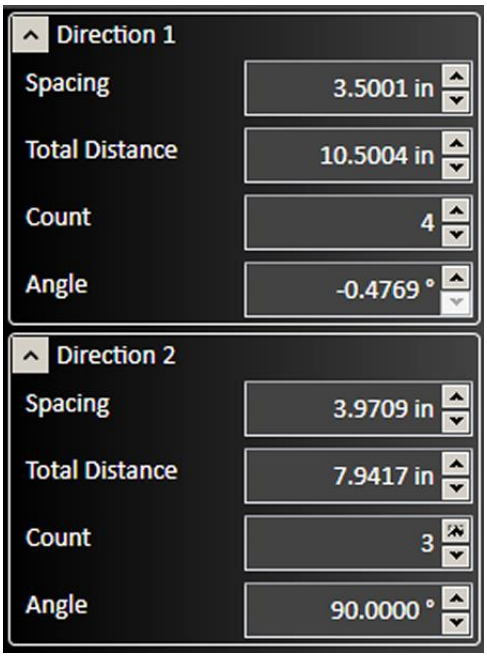
The linear pattern tool duplicates selected features in a grid fashion. Select the feature to be duplicated, select the **Linear pattern** tool then specify parameters.

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

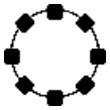
- **Direction 1** represents the horizontal direction.
- **Direction 2** represents the vertical direction.
- **Spacing** sets the distance between copies.
- **Total Distance** represents the overall distance from the original to the final copy.
- **Count** indicates the number of copies to generate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



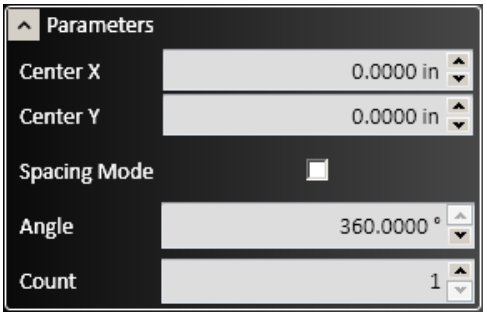




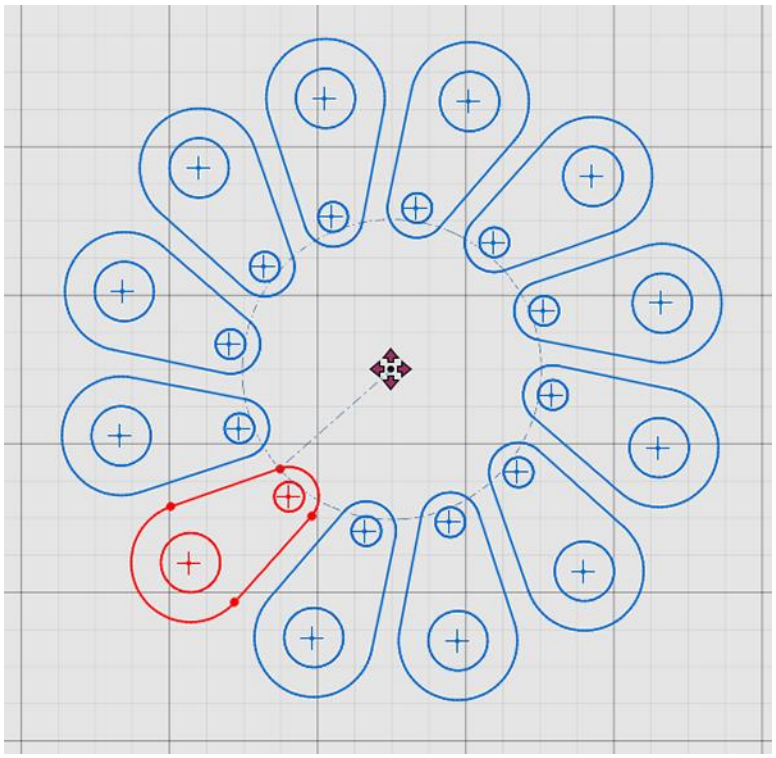
Circular Pattern Tool

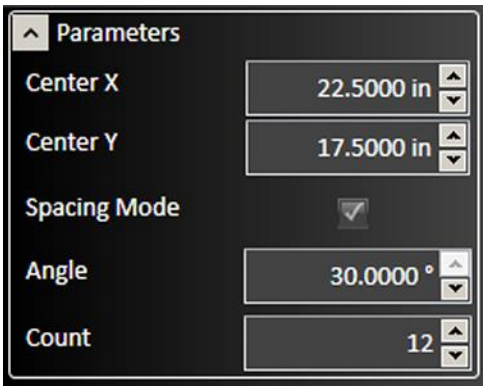


The circular pattern tool duplicates selected features in a circle around a specified point. Select the feature to be duplicated, select the **Circular Pattern** tool, and then specify theparameters.



- 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.





Pan and zoom tools

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

Pan

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

Zoom

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

Zoom 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 to Fit

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

Snap tools

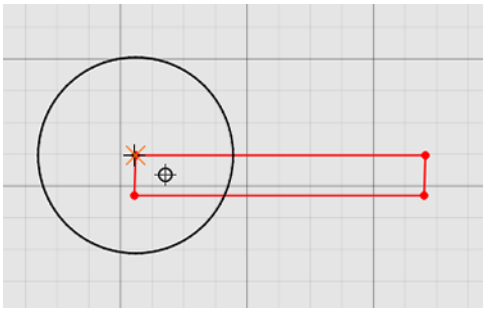
The snap tools affect how drawing tools interact with the background grid or with features already created. Clicking a snap tool toggles it on or off. Snap tools cause a feature or segment to lock on to the closest active snap location when the cursor is dragged nearby. Note that more than one type of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

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

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

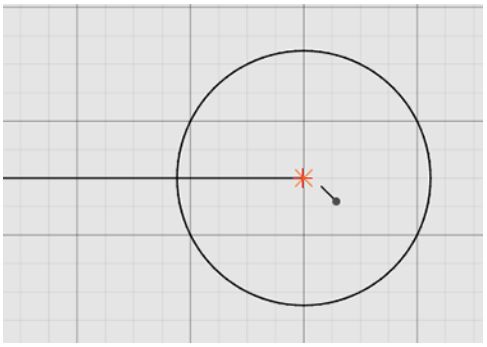
Center snap

Features connect to the centers of elements.

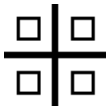


Endpoint snap

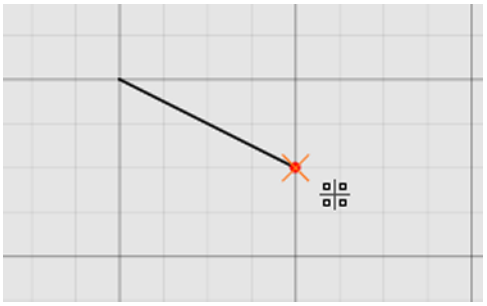
Features connect to the endpoints of elements.



Grid snap

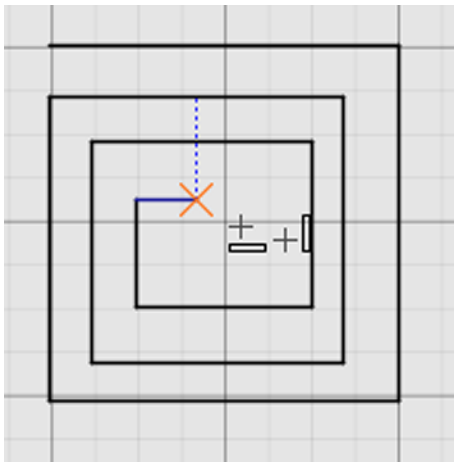


Features connect to points located along the background gridlines.



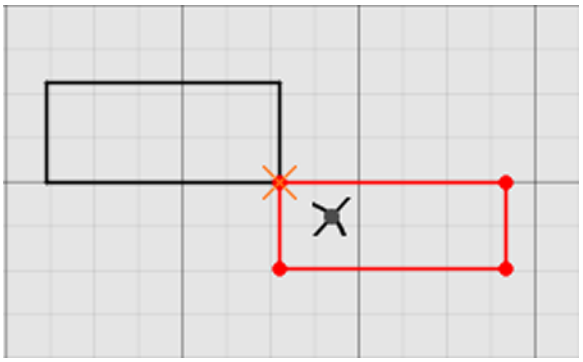
Horizontal/vertical snap

Features align vertically and horizontally when pulled into either direction.



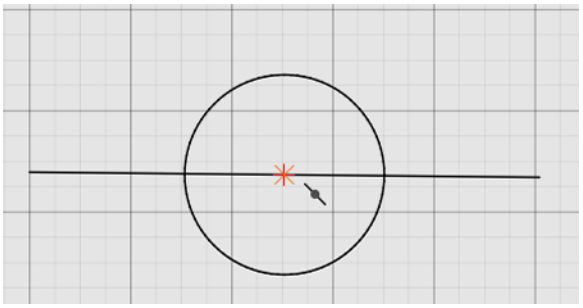
Intersection snap

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



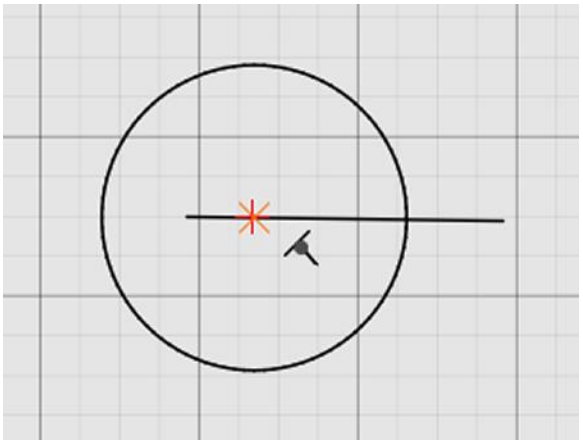
Midpoint snap

Features connect to the midpoints of elements.



Nearest snap

Features connect to the element closest to the cursor.



Parallel snap

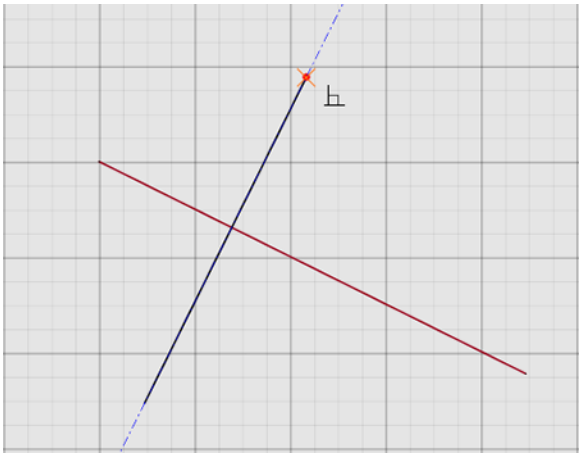
Features snap to a line parallel to the selected element.

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

Perpendicular snap

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

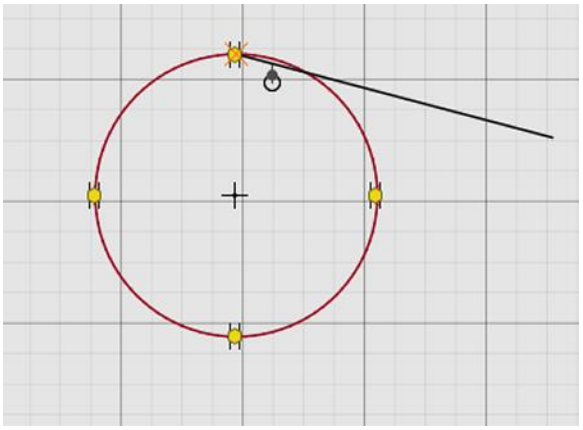
Right: a blue dashed line indicates that the line is perpendicular to the red highlighted line in the drawing.



Quadrant snap

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

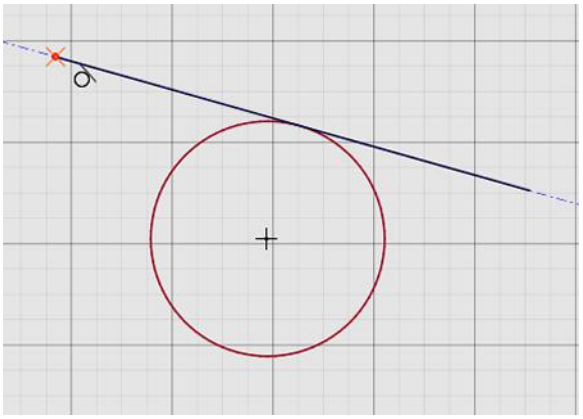
Right: line connected to a point at the top of the circle.



Tangent snap

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

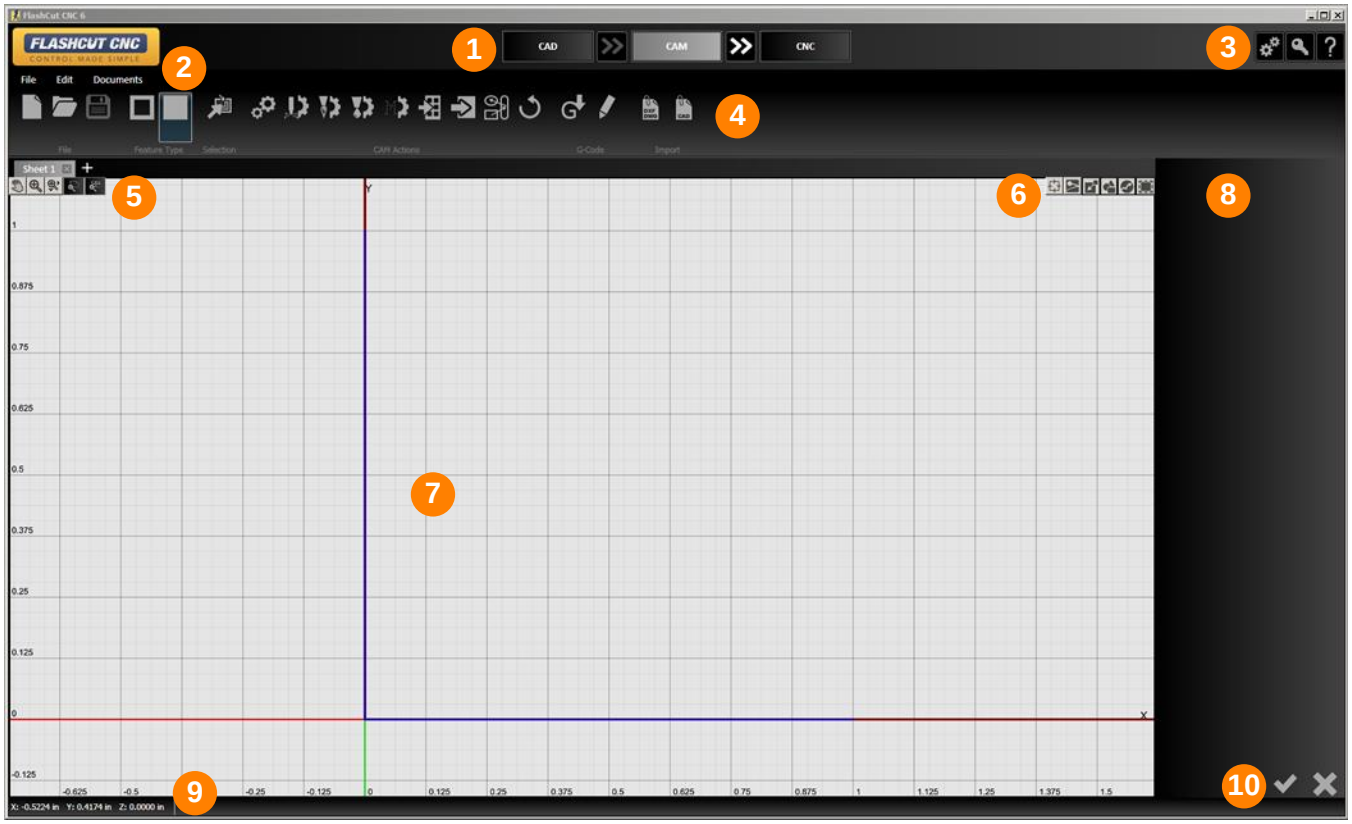
Right: a blue dashed construction line indicates that the line is tangential to the circle.



FlashCut CAM

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

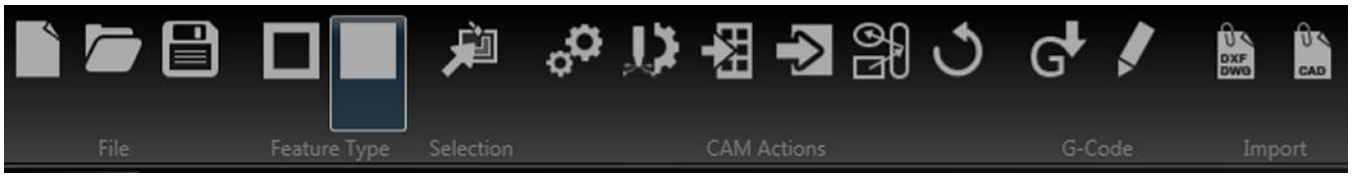
The FlashCut CAM main screen is shown here. An explanation of each area of the screen follows.



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

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

Ribbon



The FlashCut CAM ribbon has seven 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.

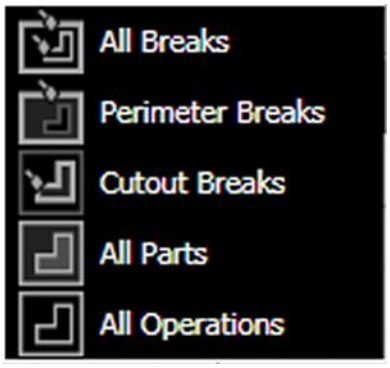
Note This option is selected by default.

Selection

When you click the **Selection** icon, FlashCut pops up the following choices for selection options: This tool allows you to select various objects in the drawing by type.

Note Breaks are the points on the toolpath where any lead in and lead outs will be placed and is represented by a red diamond on the toolpath.

For a discussion of breaks, see **Lead in/out settings**



All Breaks

Selects all available breaks in the document, including perimeter breaks and cutout breaks.

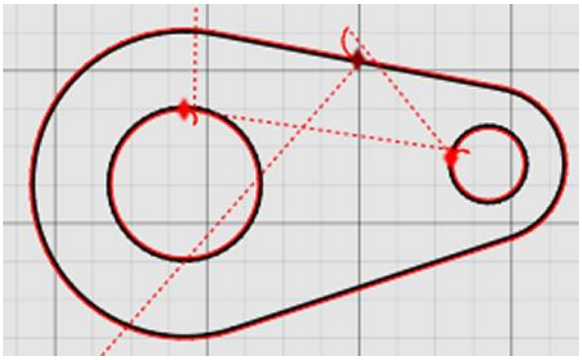
Perimeter Breaks

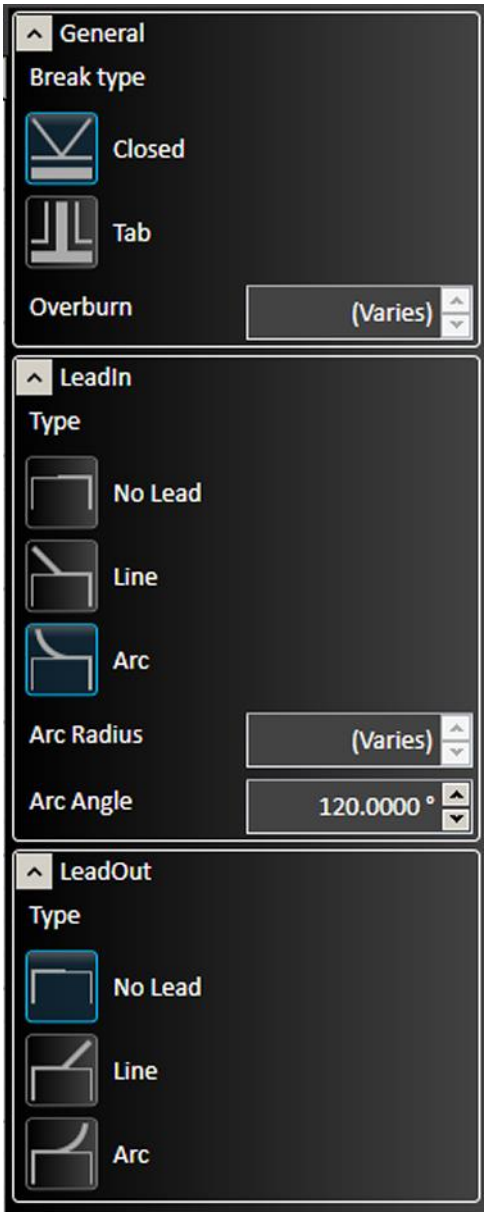
Select this option to select all breaks that are on the outside of the part. When the toolpath is offset to the outside of the part the break on this toolpath is defined as a perimeter break.

Cutout Breaks

Select this option to select all breaks that are on the inside of a cutout. When the toolpath is offset to the inside of a feature or a cutout, the break is called a cutout break.

In this example (below), the break for each circular cutout is highlighted, while the perimeter break for the surrounding part is not.





All Parts

Choose this option to select all of the parts in the viewport. This allows the user to drag and drop all parts to another position on the workpiece.

All Operations

Choose this option to select all of the operations used to fabricate the part.

Parameters

Settings for breaks are shown in the parameters window (right). For a discussion of breaks, see **Lead in/out settings**

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.

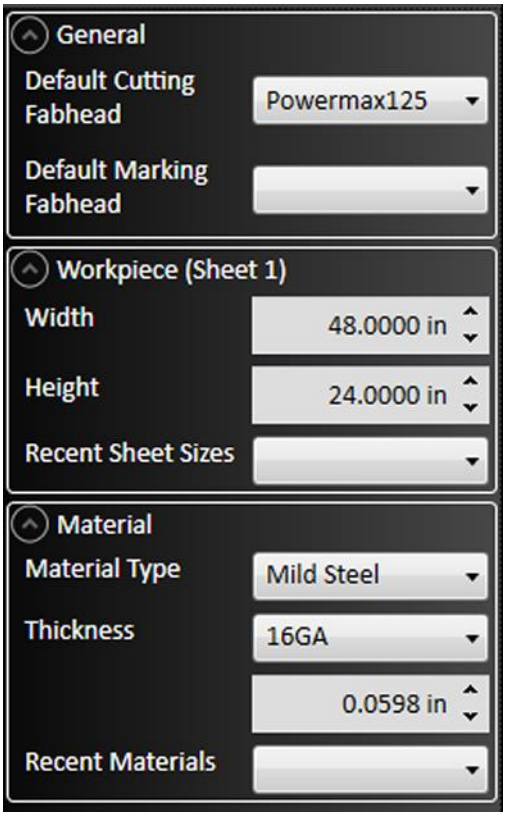
These are the tools that are available on the **CAM Actions** portion of the ribbon:

- Project Settings
- Plasma Settings
- Oxyfuel Settings
- Waterjet Settings
- Marker Settings
- Grid Nesting
- TrueShape Nesting
- Sequence tool
- CAM Reset

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.

Height

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

Recent Sheet Sizes

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

Material

Material Type

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

Thickness

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

Recent Materials

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

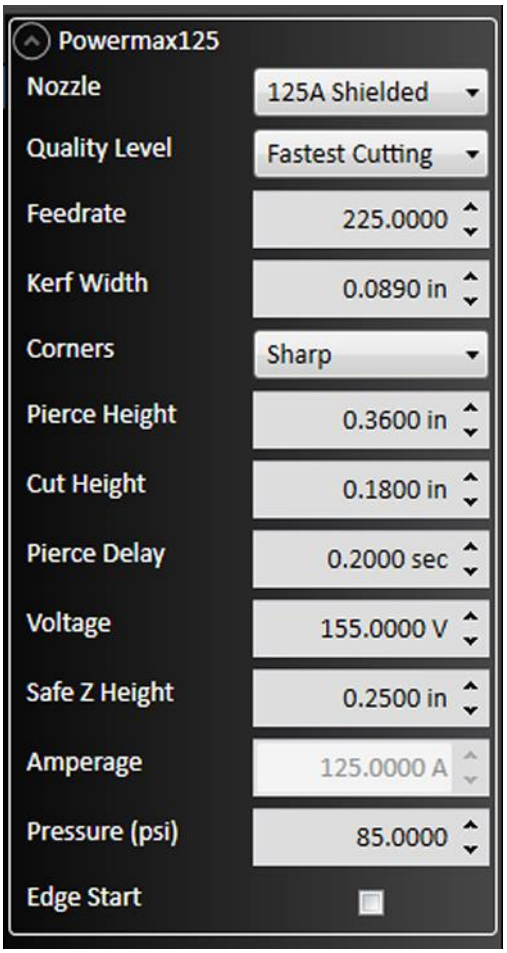
Plasma Settings

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.

Feedrate

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

Kerf Width

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

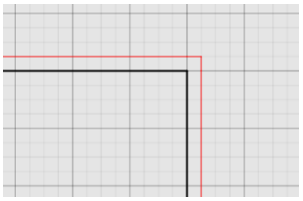
Corners

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.

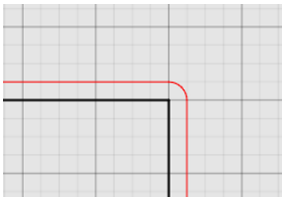
- **Sharp corners**

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



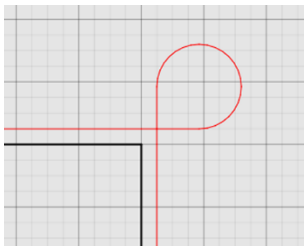
- **Rounded corners**

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



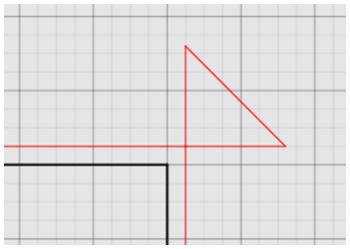
- **Rounded loops**

  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.



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.

Safe Z Height

Input the height to which the torch will return after each cut. It is a safe plane on which the torch can move, and indicates the lowest height at which the torch will make rapid moves between cutting features or parts. This is relative to the Program Zero, and is independent of the last touch off position.

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.

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.

Oxyfuel Settings

If an oxyfuel fab head has been configured, its settings can be adjusted here.

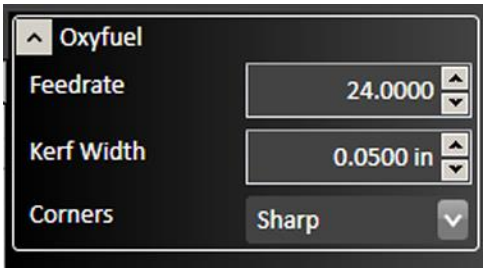
FlashCut offers a number of different options for calibrating the CAM editor to match the specifications of various fab heads. Click the icon to bring up options in the parameters window.

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

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

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

For an example, see **Corners** in the section on Plasma fab heads, above.

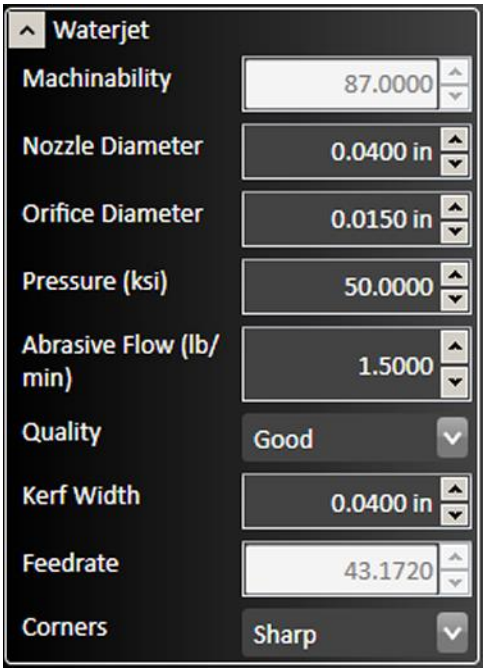
Waterjet Settings

If a waterjet 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.

Pressure (ksi)

Cutting head pressure, thousands of pounds (kilopounds) per square inch.

Abrasive Flow

Flow rate for the abrasive compound.

Quality

Low, fair, average, good, or excellent.

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.

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.

Marker Settings

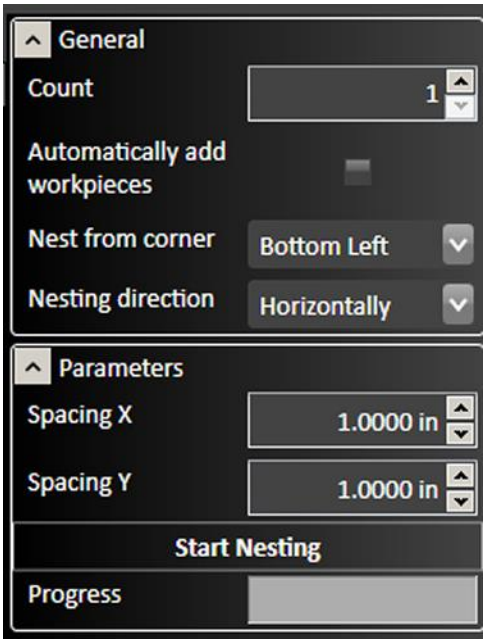
If a marker has been configured, the **Feedrate** can be adjusted here.

See **Configuring FlashCut** for more details.

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.

Spacing X

Sets the horizontal spacing between parts.

Spacing Y

Sets the vertical spacing between parts.

Start Nesting

Starts the nesting operation

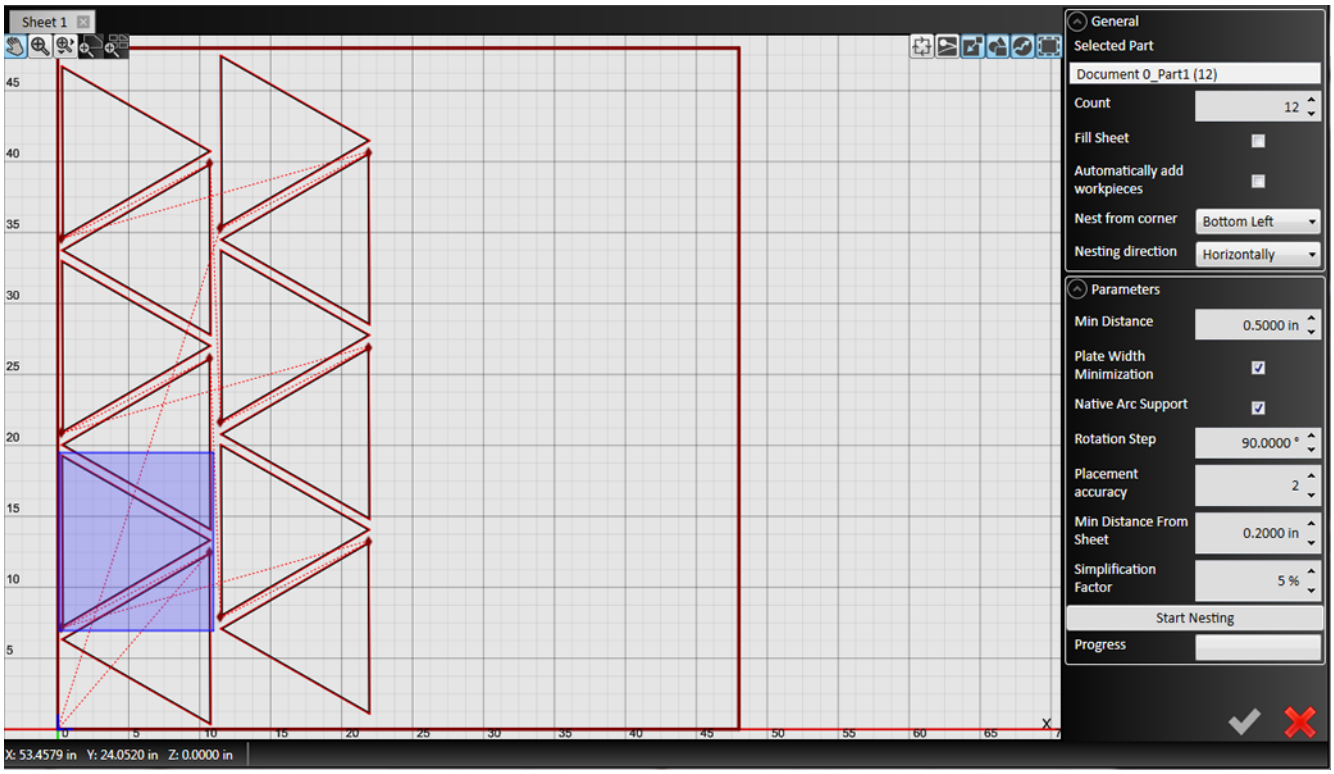
Progress

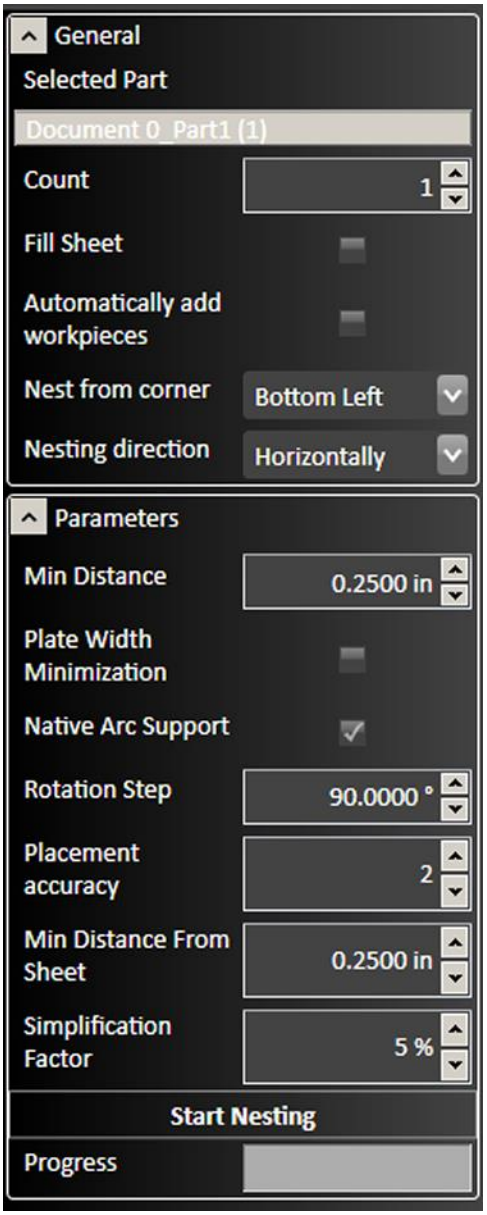
Shows the progress of the nesting operation. Complex nesting operations can take significantly longer.

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.

Min Distance

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

Plate Width Minimization

Toggle in order to minimize the horizontal space taken by the nesting shapes. Vertical arrangements will take priority if this option is toggled.

Native Arc Support

When checked, the system uses arcs in the toolpath to calculate the nest, instead of converting arcs to a linear approximation.

Rotation Step

Specify the minimum angle by which FlashCut will attempt to rotate a part when calculating the nest. By default, this setting will be 90 degrees. The smaller this step, the longer it will take for the nesting to render. Range: 0-360 degrees.

Placement accuracy

Specify the minimum step distance by which FlashCut will attempt to position a feature when calculating the nest. By default, this setting will be 0.2000 inches. The smaller the step, the longer it will take for the nesting to render. Range: 1-10.

Min Distance from Sheet

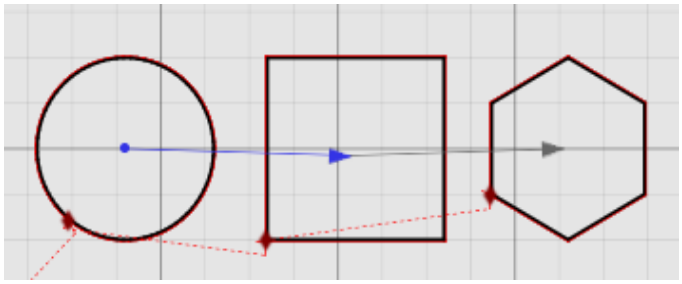
Determines the minimum distance from the edge of the sheet to any feature on any part including kerf compensation, lead ins and lead outs. A value of zero corresponds to the edge of the material.

Simplification factor

Speeds up nesting when nesting complex shapes. Complex shapes can take a long time to nest. Simplification modifies the variables sent to the nesting algorithm to balance faster nesting against perfection of spacing. A higher value will make the nesting algorithm finish faster at the cost of some precision of spacing. Range: 0-100%.

Sequence tool

When selected, the sequence tool will display the order in which parts will be cut by displaying arrows. To change the order, click the edge of a feature and then drag a blue arrow to the edge of the next feature you want cut. When finished, select the green check mark to finalize changes.



CAM Reset

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

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

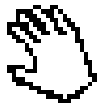
Imports DXF/DWG file to the CAM drawing workspace. The file selection dialog, and other options are shown in the **Parameters** window.

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

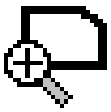
Click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window allows you to pan without selecting the pan tool first.

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.



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 to you to decide what appears in the drawing window. Click the buttons to toggle each option on and off. You may toggle any number of options at one time.

Cutting Directions

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

Kerf

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

Material

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

Parts

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

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

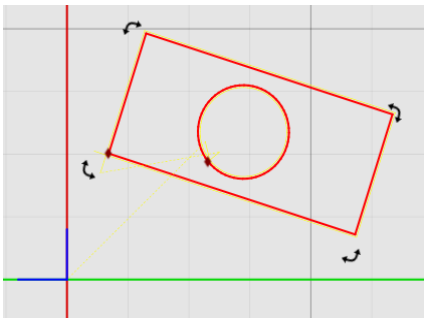
When toggled, the path of the cutting tool is outlined in red on the CAM drawing.

Drawing workspace

This section discusses the following topics:

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

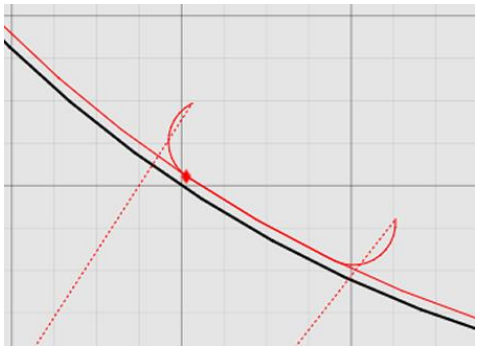
Feature manipulation

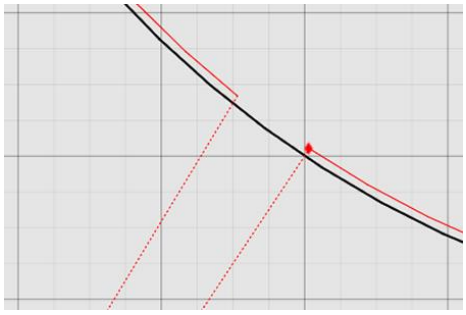


You can maneuver parts in the drawing window into a desired rotation or position. Move the cursor over the part so that it highlights green and select it so it turns red.

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.

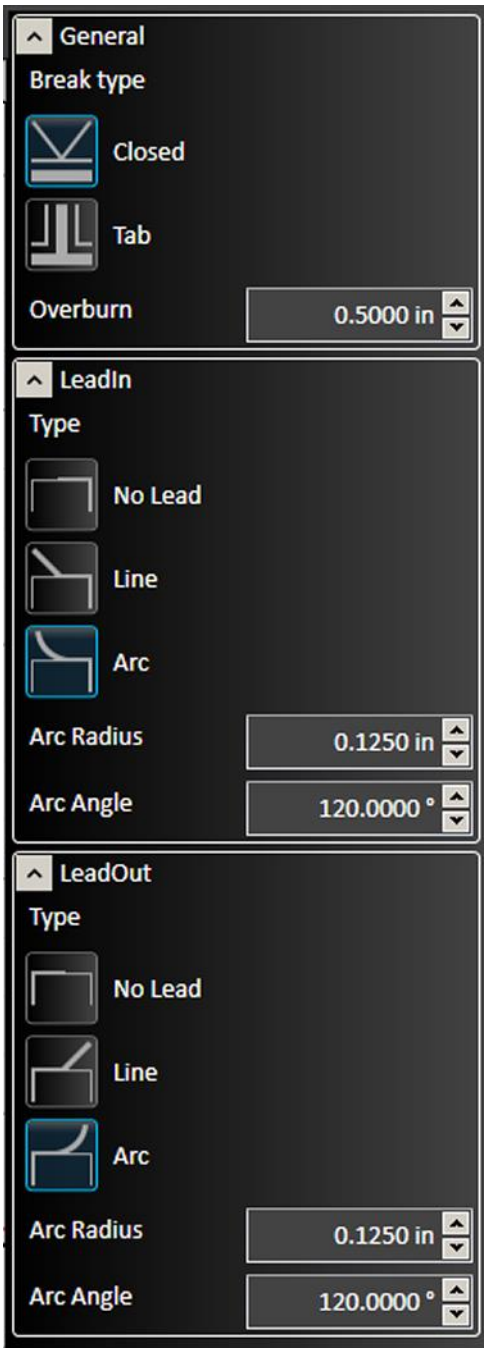
To adjust the lead in and lead out settings, select the red diamond icon on the desired feature. This brings up the parameters window.

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.

Break Type



Closed

When selected, the lead point and the end of the cut will overlap, closing the break between the beginning and end of the cut.

Overburn

To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.

Tab

When selected, there will be a gap between the lead point and end of the cut, creating a tab break between the beginning and end of the cut.

Width

The distance of the tab can be set here.

LeadIn/LeadOut Type

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

- **No Lead** removes the respective line.
- Line results in a straight line. If you choose **Line**, the **Line Length** and **Line Angle** options below are displayed.
- **Arc** creates an arc tangent to the feature. If you choose Arc, the **Arc Radius** and **Arc Angle** options are displayed.

Line Length

Select the length of the leading line.

Line Angle

Select the angle that the leading line makes with the toolpath. The angle is measured clockwise.

Arc Radius

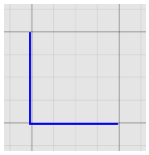
Select the radius of the circular arc.

Arc Angle

Select the angle that the arc travels through, measured from the edge of the toolpath.

Program zero

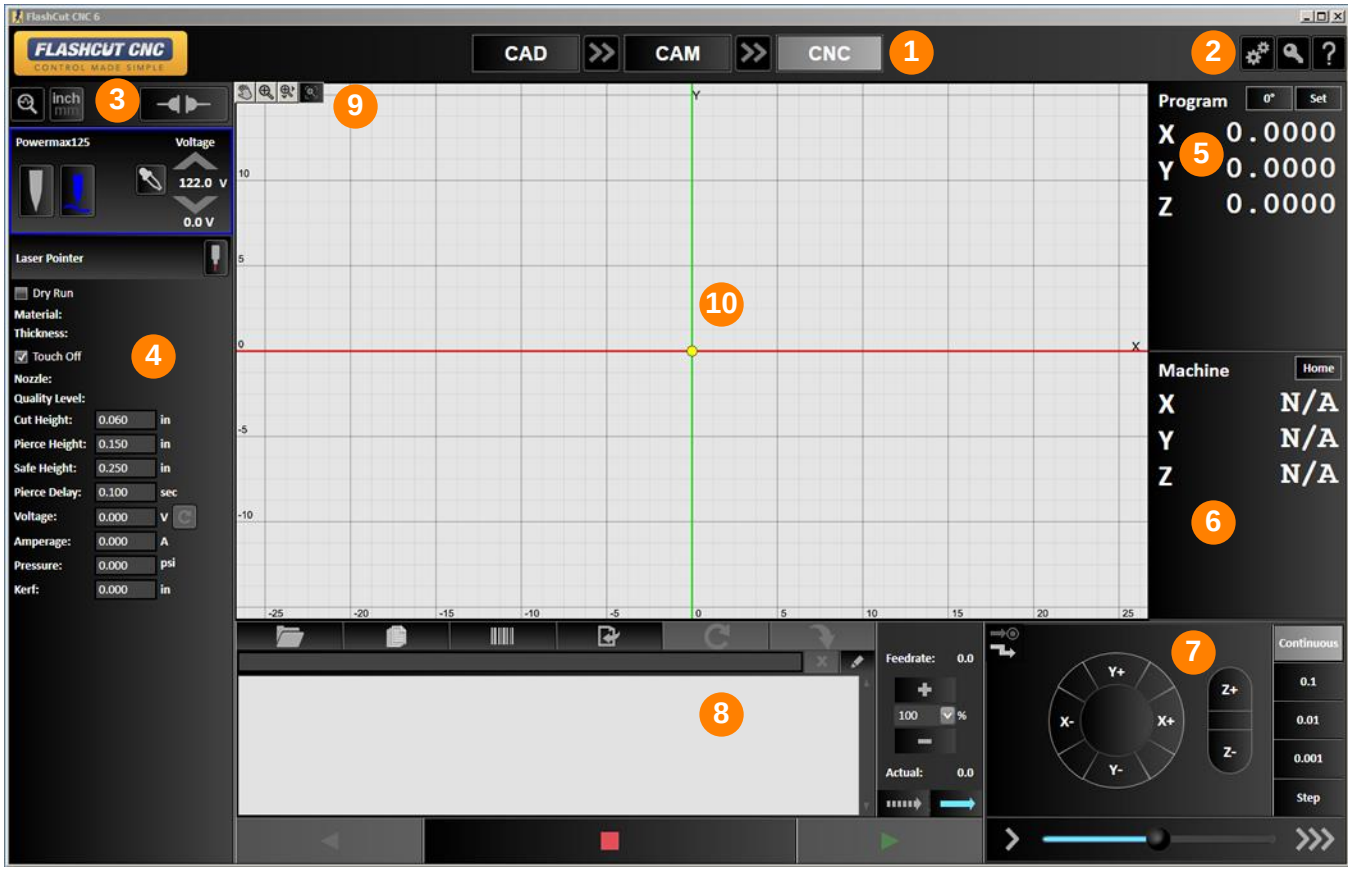
The blue set of axes in the drawing window represents the origin of the program coordinates. You may move these axes around by clicking and dragging the blue lines, or by entering coordinates manually in the parameter window. For more information on program coordinates, see **Program coordinates panel**.



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:

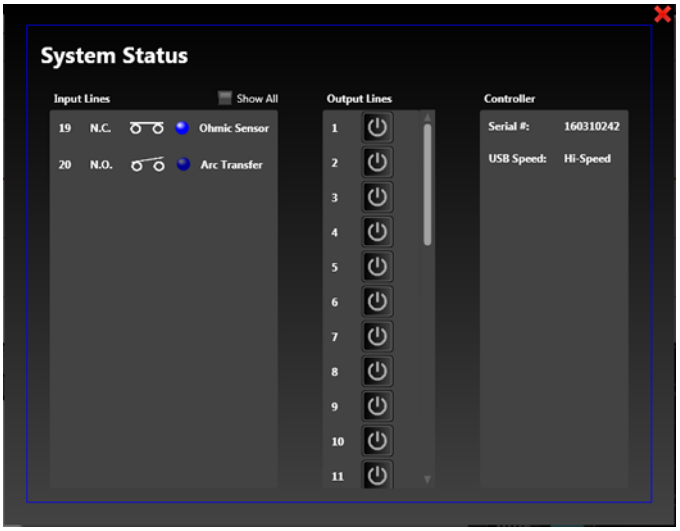
1. Tabs
2. Configuration, License, and Help buttons
3. System Status
   Toggle Display Units
   Connect
4. Fab head settings
5. Program coordinates panel
6. Machine Coordinates Panel
7. Jog and point control panel
8. G-Code window
9. Pan and zoom tools
10. Drawing workspace

System Status, Toggle Display Units, and Connect

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

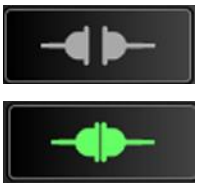
The serial number and USB speed of the controller are also shown.

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.

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.

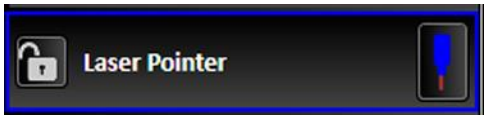
Before the unit connects, a safety reminder screen appears. It is imperative that you and anyone else near the machine understand, agree with and adhere to all of the safety guidelines. If the safety guidelines are not accepted, the software will not connect.

Fab head settings

Each fab head that has been configured and saved will be displayed here. See **Configuring FlashCut** for more information.

Laser pointer control

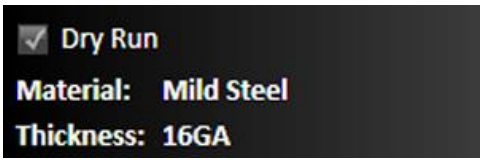
If a laser pointer is configured, it can be turned on and off by clicking the icon. The pointer can also be locked in the on position.



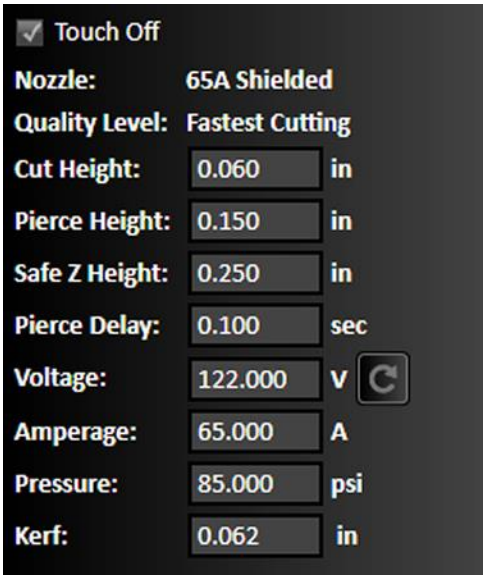
Plasma torch settings

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.

- **Torch On/Off** – Click to turn the torch on or off. Before turning on the torch, FlashCut CNC will display a warning dialog asking if it is safe to proceed.
- **Torch Height Control** – Click to turn THC on or off.
- **Use Sampling** – Click to turn sampling on or off.



- **Voltage** – Click up or down arrows to raise or lower the voltage.
- **Dry Run** – When selected, the machine will execute the operations, but the torch will not be lit, and cuts will not be made.
- **Material** and **Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.

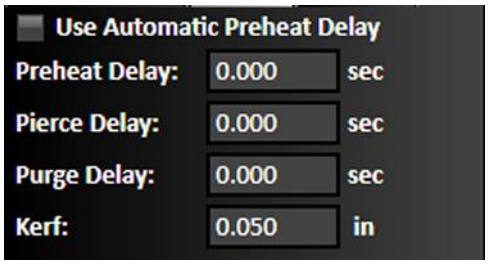


- **Touch Off** – The torch will touch the material in order to establish a zero point on the machine coordinates. Toggling **Touch Off** on will make the torch touch down to reestablish the zero point between rapid and feedrate moves.
- **Nozzle** – Displays information about the plasma torch nozzle.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

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.

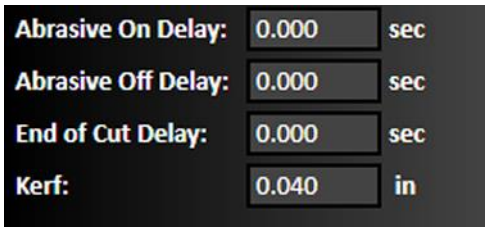
- **Use Automatic Preheat Delay** – Select to turn on this feature.
- **Preheat Delay** – Specifies the time to preheat the material before cutting.
- **Pierce Delay** – Specifies the time to delay before completing a pierce operation.
- **Purge Delay** – Specifies the time to delay before purging the cutter.
- **Kerf** – Specifies the width of the kerf used by the cutter.



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.

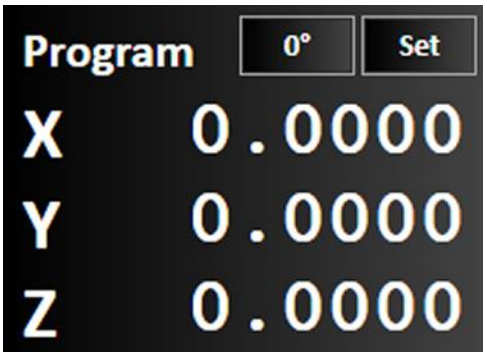
- **Abrasive On Delay** – Specifies the time in seconds before the flow of abrasive starts after the cutter is activated.
- **Abrasive Off Delay** – Specifies the time before the flow of abrasive ends when the cutter is deactivated.
- **End of Cut Delay** – Specifies the delay at the end of the cut. The purpose of the delay is to make sure the cut is complete.
- **Kerf** – Specifies the width of the kerf used by the cutter.



Marker control

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

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.
- **Sense Lift Axis Zero** instructs the plasma torch to touch down onto the material to be cut. This enables FlashCut to determine how high the torch will remain while it cuts the material.
- **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

Coordinate rotation example

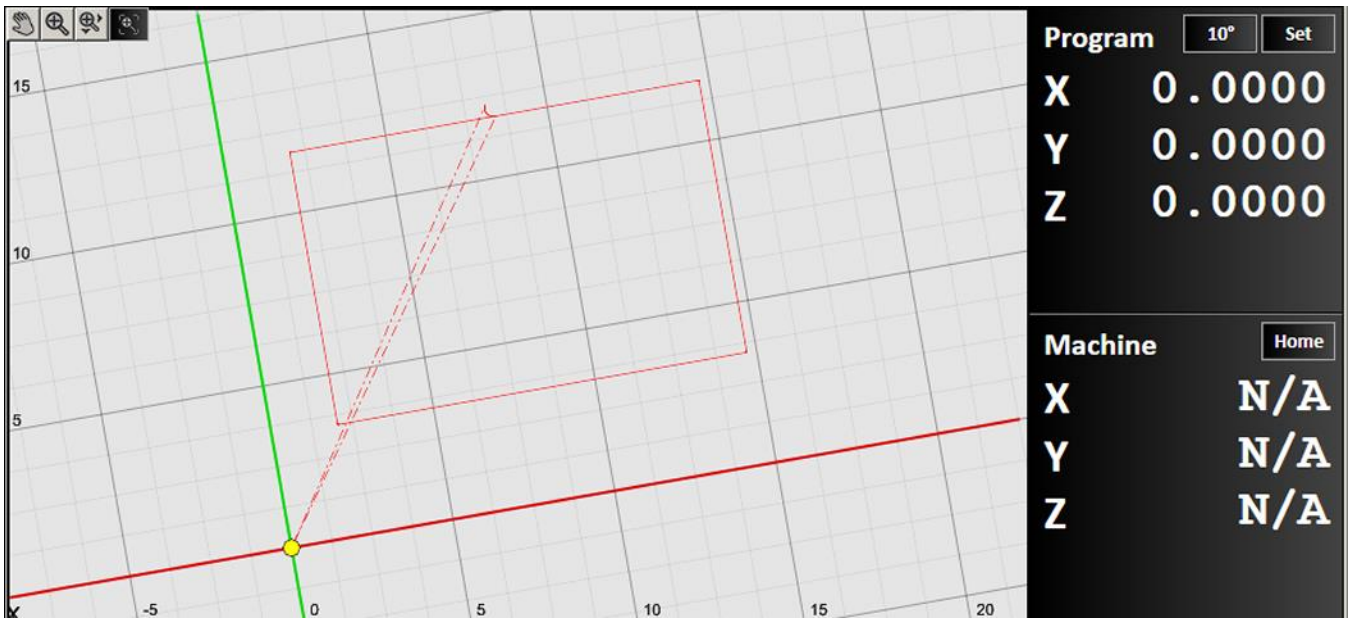
Coordinates rotation can be used to accommodate a workpiece that is too heavy or inconvenient to move on the machine. In effect, the program axes can be adjusted to conform to the dimensions and orientation of the workpiece.

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.

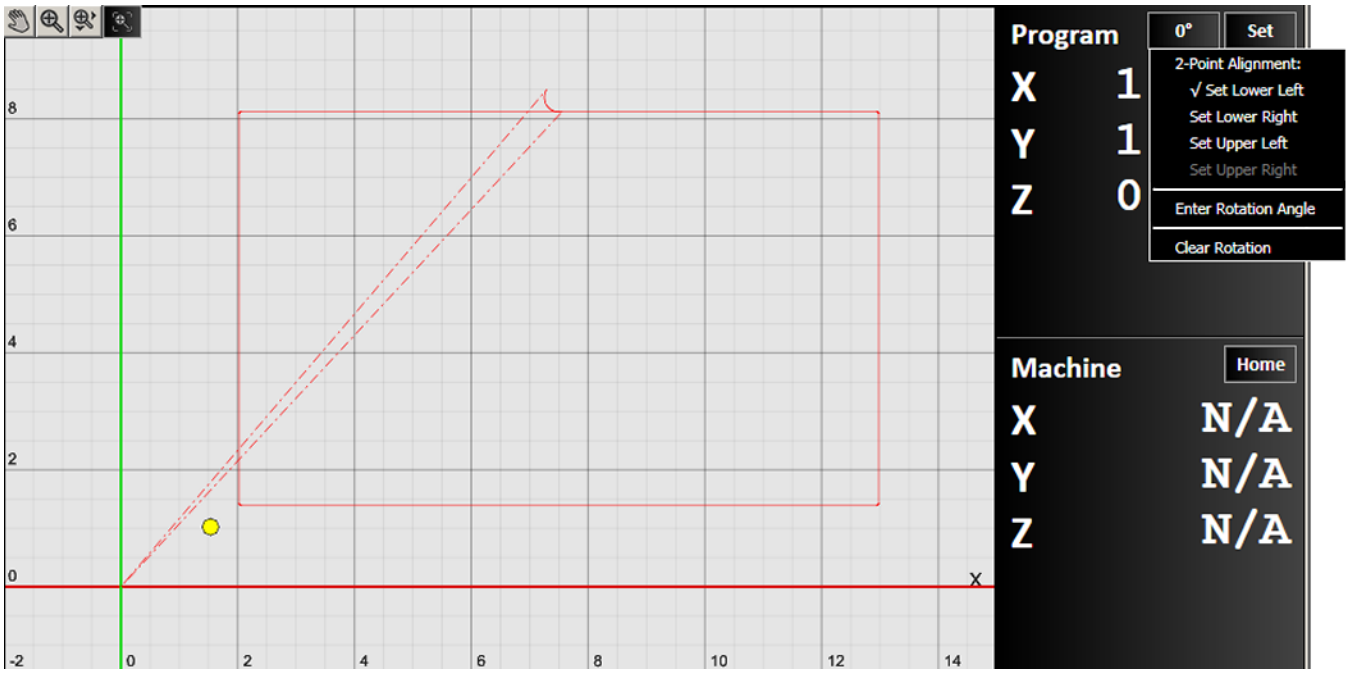
The angle of rotation will be displayed in the Program coordinates panel, and the drawing workspace will be rotated.



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.

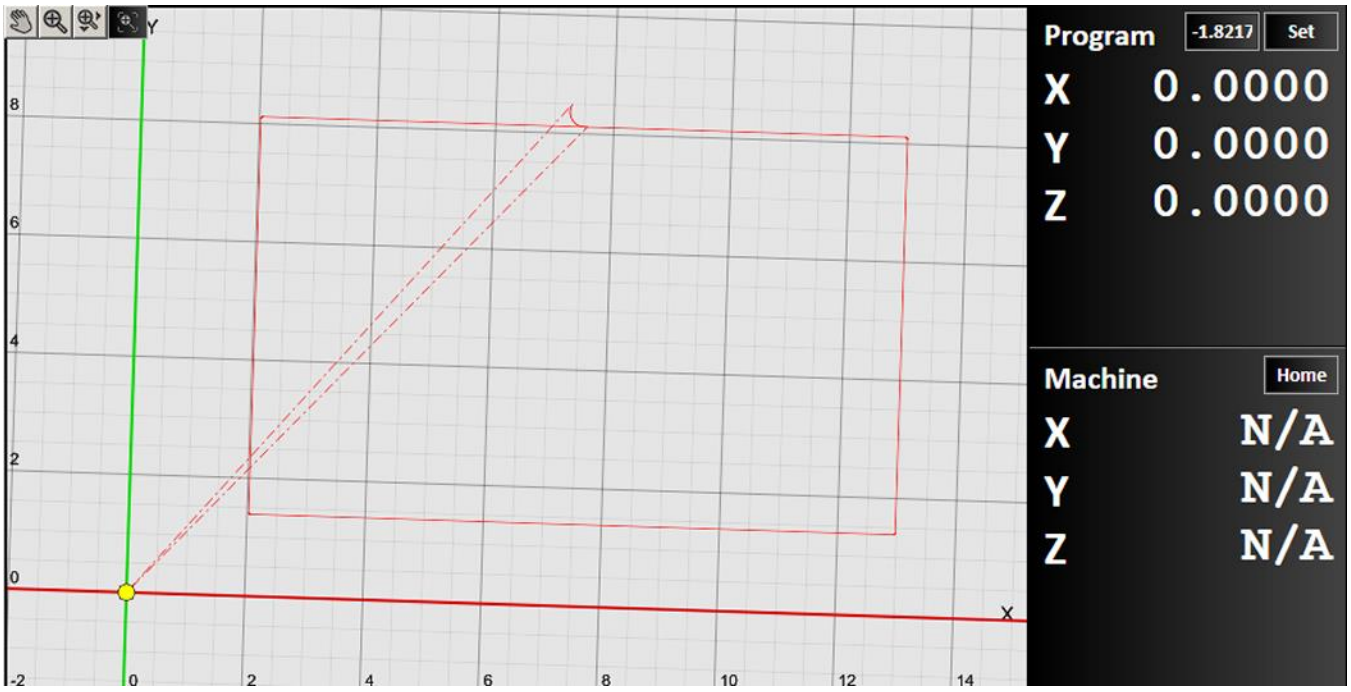
1. Jog the tool position to the lower right corner of the workpiece
2. Select **Set Lower Left** from the dropdown menu



3. Jog the tool position to the lower right corner of the workpiece

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

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



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

Machine Coordinates Panel

The machine coordinates refer to the exact position of the tool with respect to the machine home. The coordinates will read **N/A** until the tool is able to locate its home position.

After connecting the signal generator to FlashCut, it is necessary to seek out and set the home point, or **Machine Zero**, of the tool. It is recommended that each axis is jogged first near the home switch before homing. Once machine zero is set, the machine tool envelope is redefined in the workspace.

The **Home** dropdown menu provides the following commands:

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

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

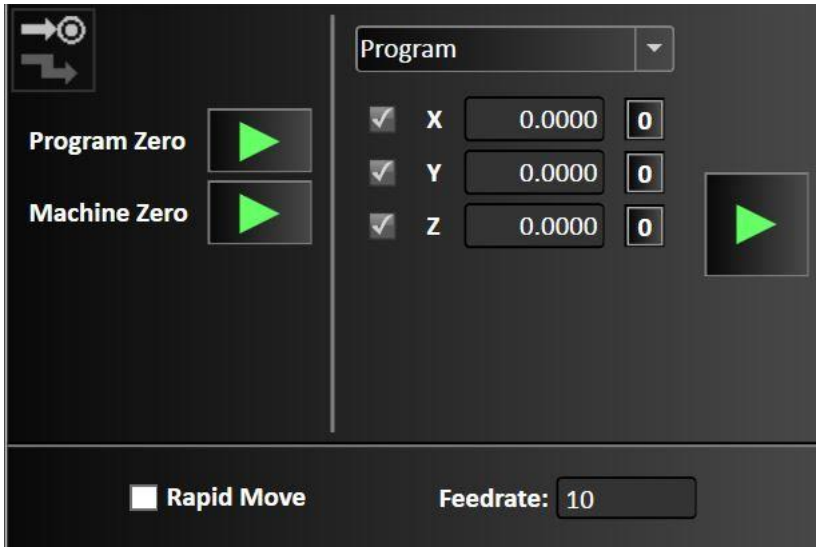
The jog control panel allows you to move your tool location in two ways.

- Move to point
- Jogging

This icon in the top left corner of the jog control panel is a button that toggles between them.

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. If you prefer the move to be made at the maximum allowable speed, check the **Rapid Move** box.



- 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.

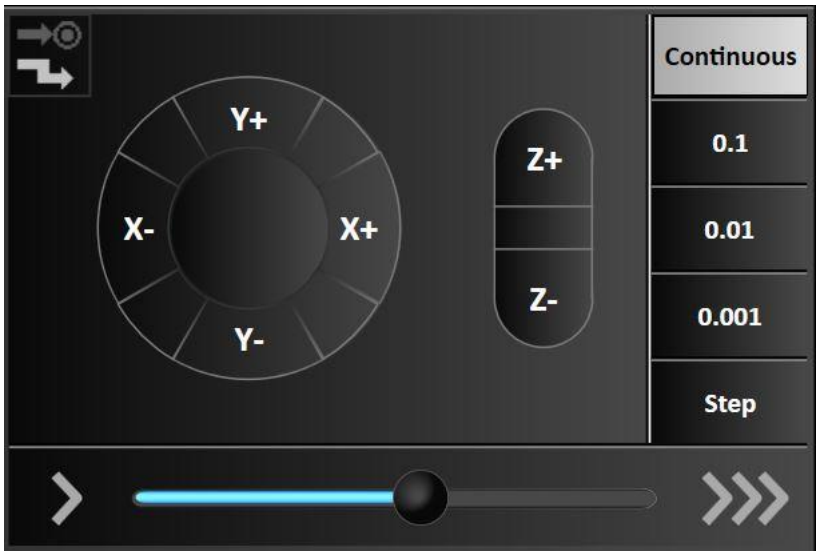
You may also use the controls on the right side of the jog-to–point panel to modify the destination based on your selection with the dropdown. The choice you make with the dropdown specifies which point the move will be made with respect to. You may choose from:

- **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.

Jogging

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



Axis jog buttons

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

Diagonal jog buttons

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

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

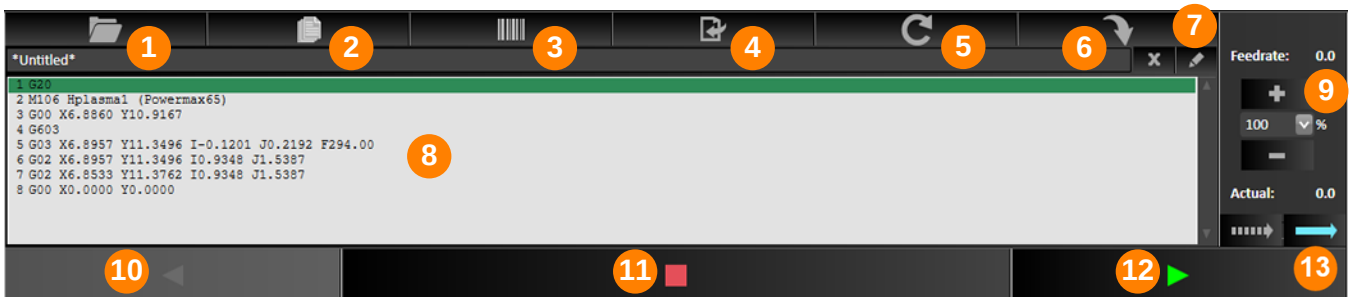
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

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.

This section describes these G-Code window features:



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

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.

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 in reverse

This button is not enabled yet, but will allow the user to run the program in reverse. This is beneficial 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.

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.

Step/Continuous mode toggle

Select whether the G-Code will be executed continuously as a single set of moves or step-by-step by executing each move line-by-line. The blue option is the one that is selected.

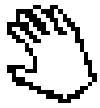
Step mode

Continuous mode

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

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 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.

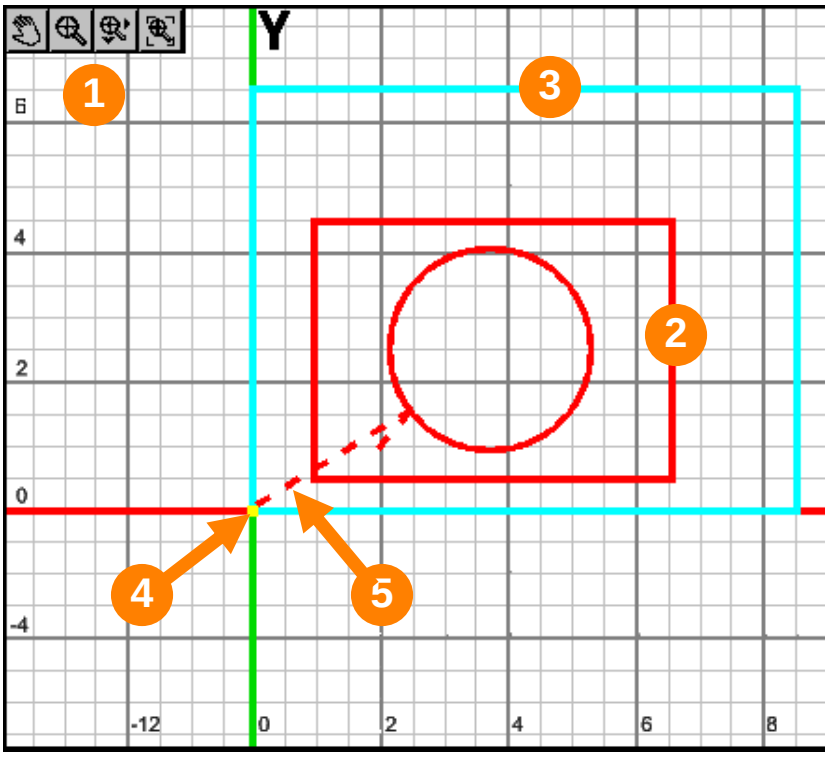
Zoom to Fit

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**
2. Solid red lines show the toolpath
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.

   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.
4. The current tool position is shown by a yellow dot.
5. Dotted red lines represent rapid moves

Configuring FlashCut

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

Note If an Administrator password has been set, you must enter it before you can make changes to the configuration. See **Security** for more information.

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:

1. **Setup file (*.setup extension)**
2. **Configuration workspace**
3. **Configuration Menu**

Setup file (*.setup extension)

The Setup file contains all of the configuration parameters for your machine tool. You can change the parameters using commands in the **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 6\Application Data\Configuration** folder but you may not have sufficient privilege to store your setup files there. Choose an alternative location.

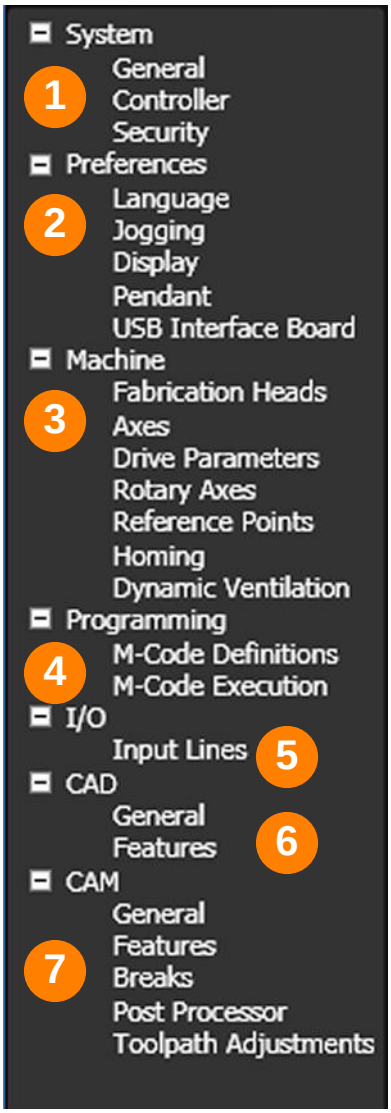
If a file has not been created yet, the **Save As...** button enables you to create a new .setup file and destination.

Configuration workspace

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

Configuration Menu

1. System
   - General
   - Controller
   - Security
2. Preferences
   - Language
   - Jogging
   - Display
   - Pendant
   - USB Interface Board
3. Machine
   - Fabrication Heads
   - Axes
   - Drive Parameters
   - Rotary Axes
   - Homing
   - Dynamic Ventilation
4. Programming
   - M-Code Definitions
   - M-Code Execution
5. I/O
   - Input Lines
6. CAD
   - General
   - Features
7. CAM
   - General
   - Features
   - Breaks
   - Post Processor
   - Toolpath Adjustments



System

- General
- Controller
- Security

General

System Units

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

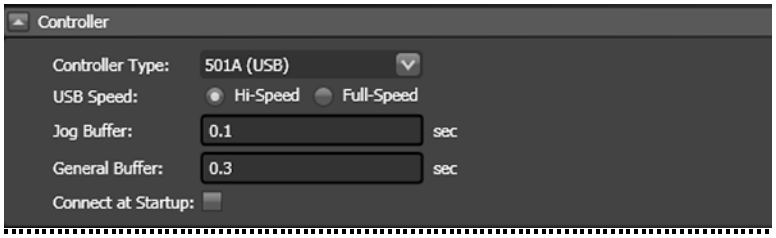
- English
- Metric

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. If your machine is exposed to excessive electrical noise, 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)

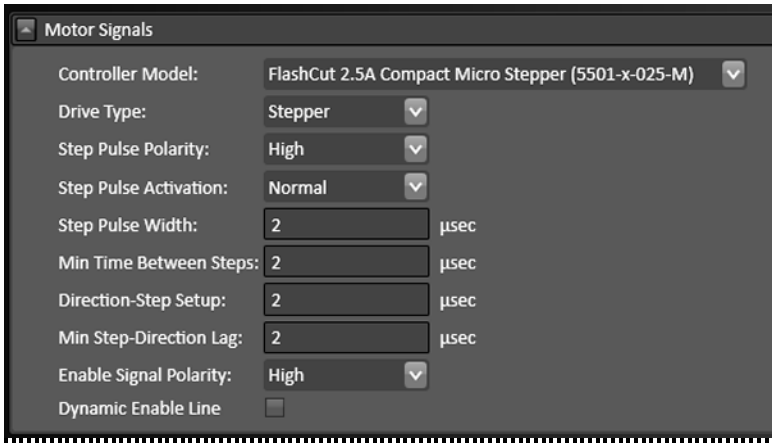
General Buffer (sec) sets the buffer size on the signal generator for all operations except jogging in continuous mode.

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



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).

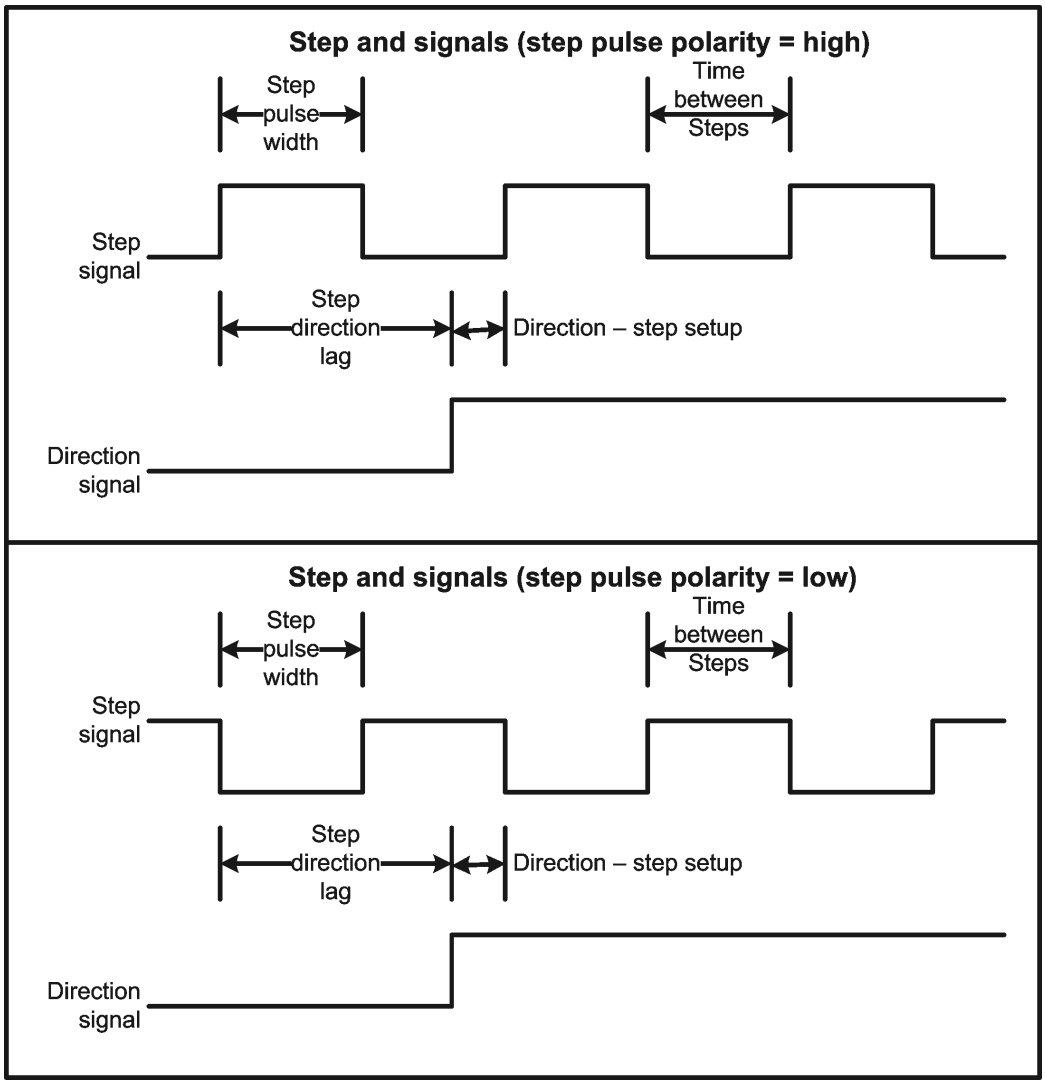
If you are using non-FlashCut drivers, choose **Other**. This enables all fields, allowing you to enter the proper settings based on your motor driver requirements. Note that some signal fields do not apply to all signal generator models, and may be disabled based on your controller settings (signal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

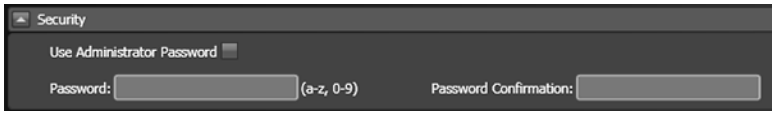
Sets the state to which the signal generator drives the motor enable line when you disconnect from the signal generator.

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.

Security

- Use Administrator Password
- Password and 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)

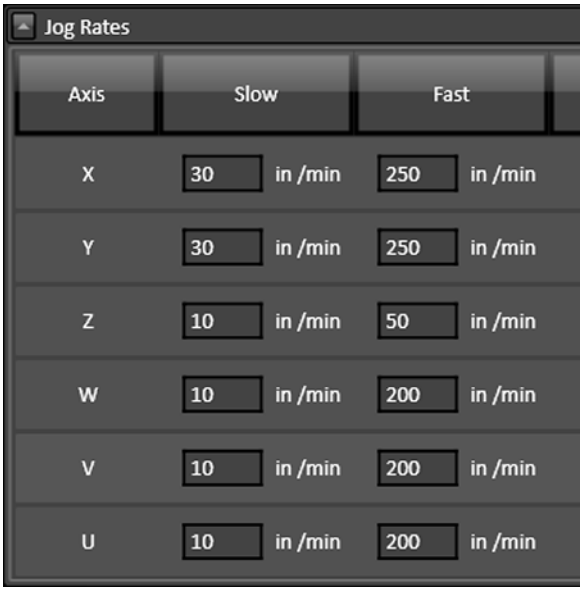
Jogging

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

- Jog Rates
- Jog Distances
- General

Jog Rates

Use the Jog Rates controls to set the default jog rates for each axis.



Slow, Fast

These columns correspond to the jog rate option buttons on the **Jog and point control panel**. FlashCut executes all jog moves using the rates you enter in these fields.

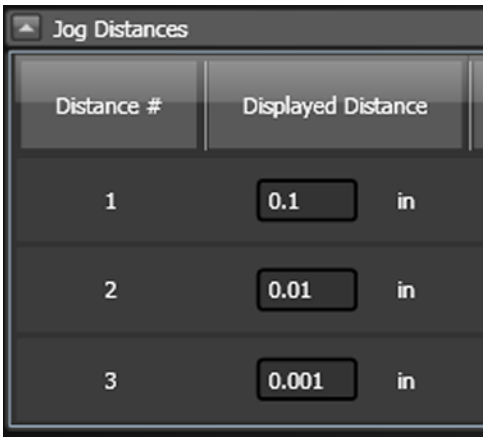
Jog Distances

Distance

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

Displayed Distance

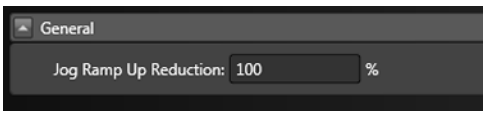
This is the value of the incremental move that is commanded each time the user clicks a directional jog button on the CNC panel.



General

Jog Ramp Up Reduction

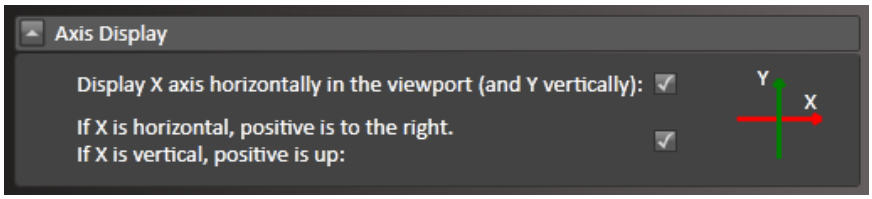
Set the percentage of the speed at which the rate increases during jog operations. The default is 100%.



Display

Axis Display

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



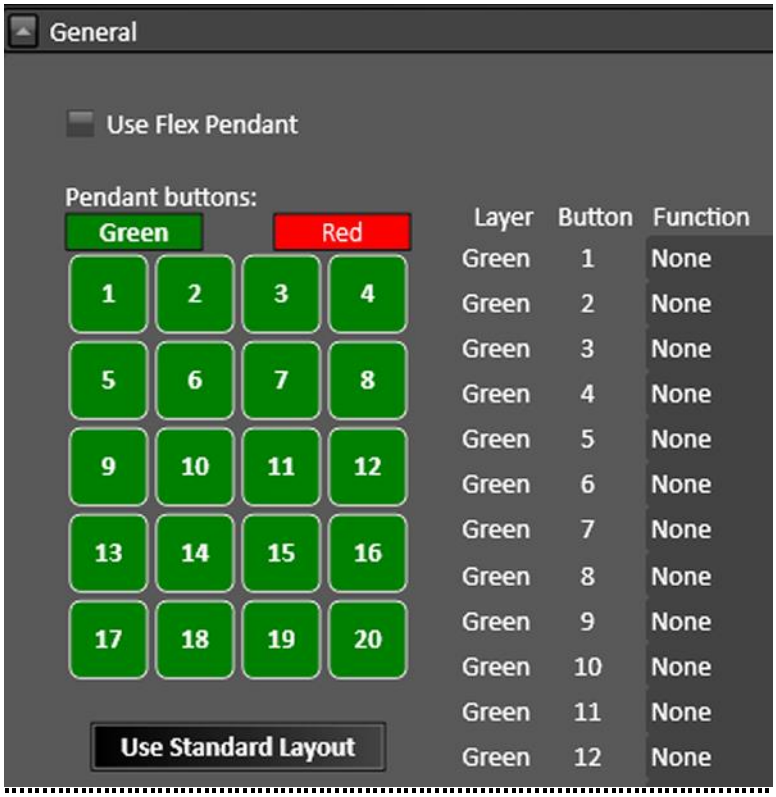
Current Position

Select the desired icon to mark the current position.

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 Flex Pendant

Enables control of the machine from the pendant.

Pendant Buttons

Two sets of 20 buttons (green and red) can be configured. Click the button you wish to configure, and then assign its function.

Use Standard Layout

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

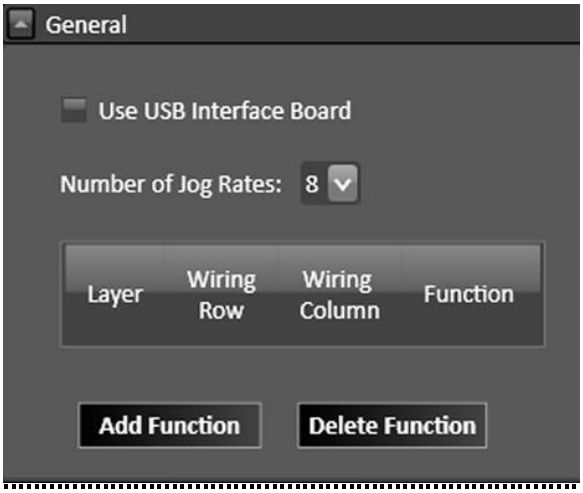
Functions

- Layer (green or red)
- Button (1-20)
- Function (select from dropdown menu)

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

Each function has four attributes:

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

Machine

- Fabrication Heads
- Axes
- Drive Parameters
- Rotary Axes
- Homing
- Dynamic Ventilation

Fabrication Heads

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

- Cut Charts
- Fabrication Heads

Cut Charts

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

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

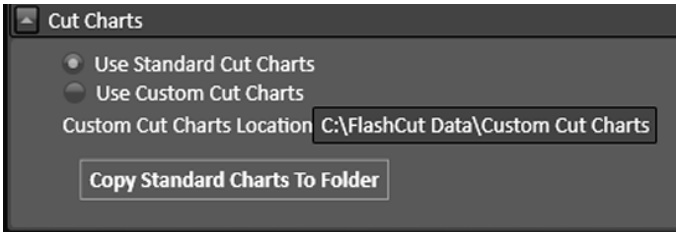
An example is shown below:

	NozzleMc	TorchLead	MaterialT	Thickness	KerfWidth	CutHeight	PierceHei	PierceDel	BestFeedi	BestVolta	Productio	Productio	Current	Pressure	EdgeStart	Units
2	125A Shie	25	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	85	N	English
3	125A Shie	50	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	85	N	English
4	125A Shie	75	Mild Steel	0.25	0.089	0.18	0.36	0.2	188	158	225	155	125	95	N	English
5	125A Shie	25	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	85	N	English
6	125A Shie	50	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	85	N	English
7	125A Shie	75	Mild Steel	0.375	0.094	0.18	0.36	0.4	114	158	138	158	125	95	N	English
8	125A Shie	25	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	85	N	English
9	125A Shie	50	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	85	N	English
10	125A Shie	75	Mild Steel	0.5	0.095	0.18	0.36	0.5	75	158	93	158	125	95	N	English
11	125A Shie	25	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	85	N	English
12	125A Shie	50	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	85	N	English
13	125A Shie	75	Mild Steel	0.625	0.103	0.18	0.45	0.6	50	162	66	164	125	95	N	English
14	125A Shie	25	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	85	N	English
15	125A Shie	50	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	85	N	English
16	125A Shie	75	Mild Steel	0.75	0.108	0.18	0.45	0.8	42	164	48	163	125	95	N	English
17	125A Shie	25	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	85	N	English
18	125A Shie	50	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	85	N	English
19	125A Shie	75	Mild Steel	0.875	0.109	0.18	0.45	2	31	168	37	166	125	95	N	English
20	125A Shie	25	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	85	N	English
21	125A Shie	50	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	85	N	English
22	125A Shie	75	Mild Steel	1	0.123	0.18	0.45	3.5	23	169	30	167	125	95	N	English
23	125A Shie	25	Mild Steel	1.25	0.15	0.18	0.18	1	16	174	20	172	125	85	Y	English

Powermax125CutChart

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.

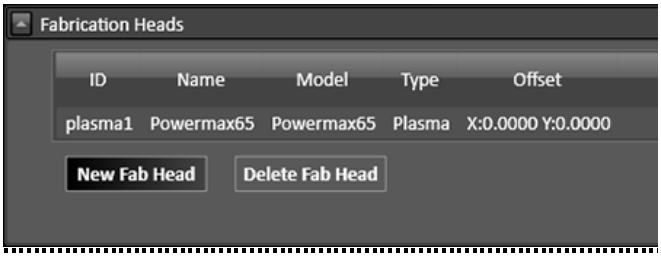
Any value in these cut charts can be edited using a spreadsheet program like Excel.

Fabrication Heads

Fabrication heads are the actual cutting heads on your machine.

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

- Laser Pointer
- Marker
- Mill
- Oxyfuel
- Waterjet
- Plasma



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.

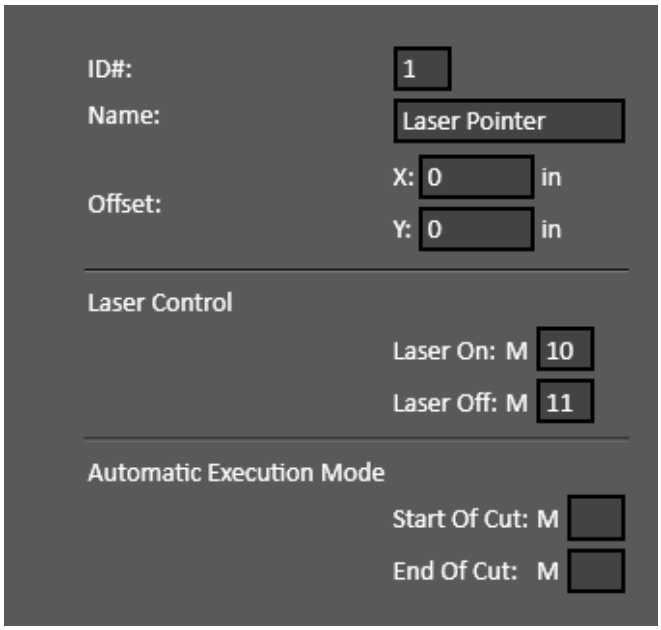
To edit a Fab Head, select it from the list and the Fab Head parameters will appear.

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

Laser Pointer

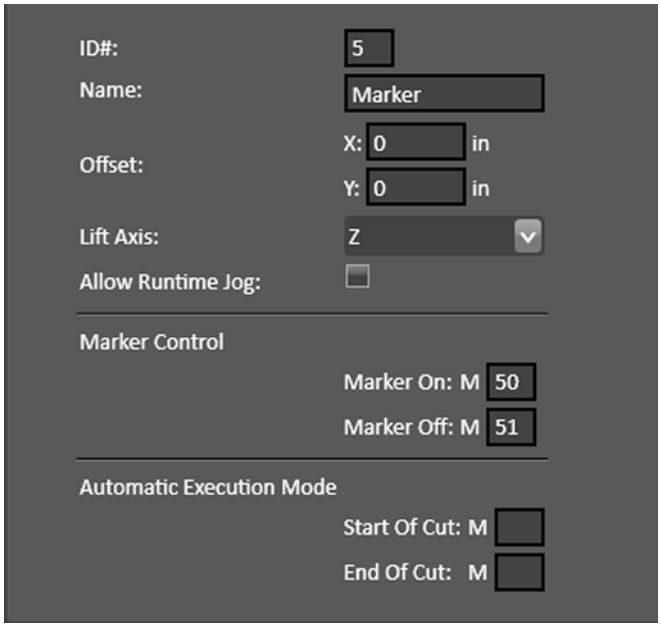
When defining a new Laser Pointer, select the **Type** from the drop down lists.

- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter the X and Y distance from the center of the laser pointer to the center of the Plasma Cutter.
- **Laser Control** – Fill in the proper M codes for turning the Laser Pointer on and off. Typically these are 10 and 11, respectively. Note that all M codes must be defined in the M-Code Definitions.
- **Automatic Execution Mode** – Since a laser pointer is a simple device, there is probably no need for a macro. Enter the same values used for Laser Control.



Marker

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



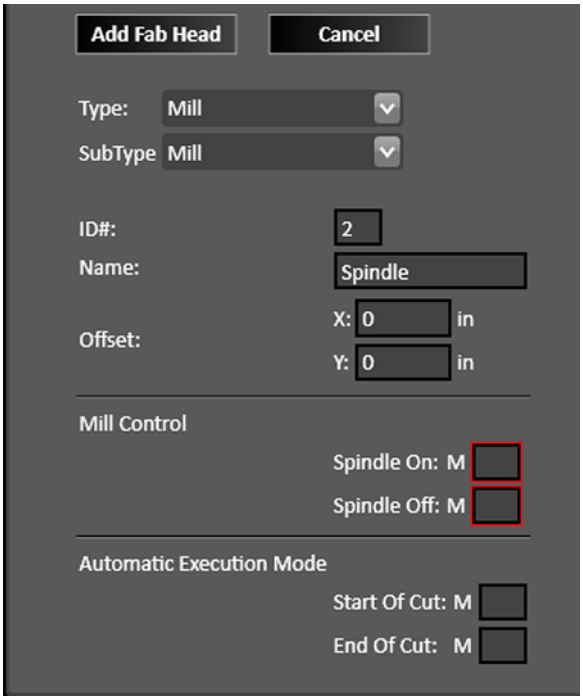
- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one fab head.
- **Lift Axis** – Select axis to be used when lifting the fab head.
- **Allow Runtime Jog** – Enables jog of lift axis. Jog is suspended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

  Marker On/Off – Fill in the proper M-Codes for turning the marker on and off. Note that the M codes must be defined in the M-Code Definitions.
- **Automatic Execution Mode**

  Start/End of Cut – Enter the Automatic M-Codes or Macros for turning the marker on for cuts and off for rapid movements. Note that the Macros and M-Codes must be defined in the M-Code Definitions. Automatic Execution must be enabled in M-Code Execution.

Mill

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

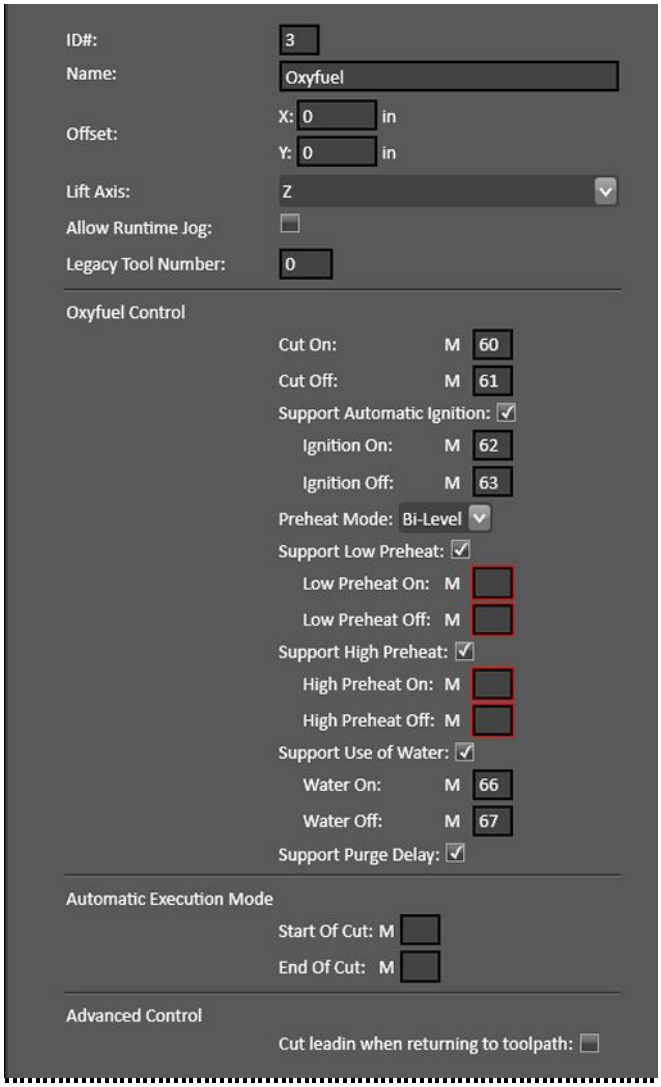


- **ID** and **Name** – These values will be automatically populated, but you can edit them.
- **Offset** – Enter an X and Y offset if you have more than one fab head.
- **Mill Control**

  Spindle On/Off – Fill in the proper M-Codes for turning the spindle on and off. Note that the M codes must be defined in the M-Code Definitions.
- **Automatic Execution Mode**
- **Start/End of Cut** – Enter the Automatic M-Codes or Macros for turning the cutter on for cuts and off for rapid movements. Note that the Macros and M-Codes must be defined in the M-Code Definitions. Automatic Execution must be enabled in M-Code Execution.

Oxyfuel

When defining a new Oxyfuel 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.
- **Legacy Tool Number** – Enter number here.
- **Oxyfuel Control** – For each of the parameters that can be turned on and off, the proper M-Codes must be defined in the M-Code Definitions table.

  Cut On/Off – Fill in the proper M-Codes for turning the cutter on and off. Typically, these are 60 and 61, respectively.

  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.

- **Automatic Execution Mode** –

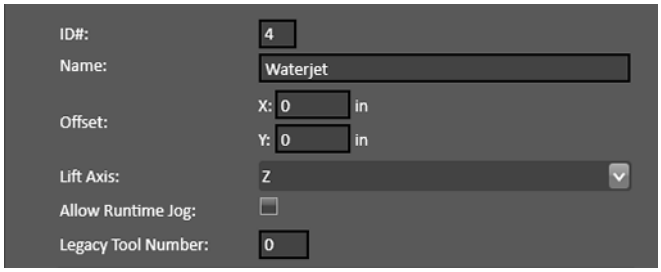
  Start/End of Cut – Enter the Automatic M-Codes or Macros for turning the cutter on for cuts and off for rapid movements. Note that the Macros and M-Codes must be defined in the M-Code Definitions. Automatic Execution must be enabled in M-Code Execution.

- **Advanced Control**

  Cut lead in when returning to toolpath – When checked, allows recovery from a lost torch.

Waterjet

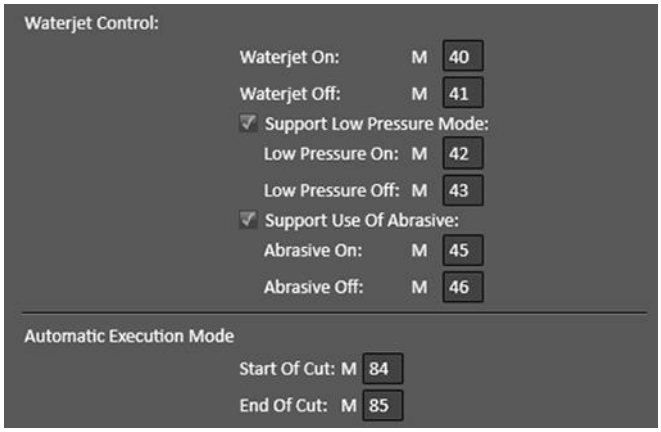
When defining a new Waterjet head, select the **Type** from the drop down lists.



- **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.
- **Legacy Tool Number** – Enter number here.
- **Waterjet Control**

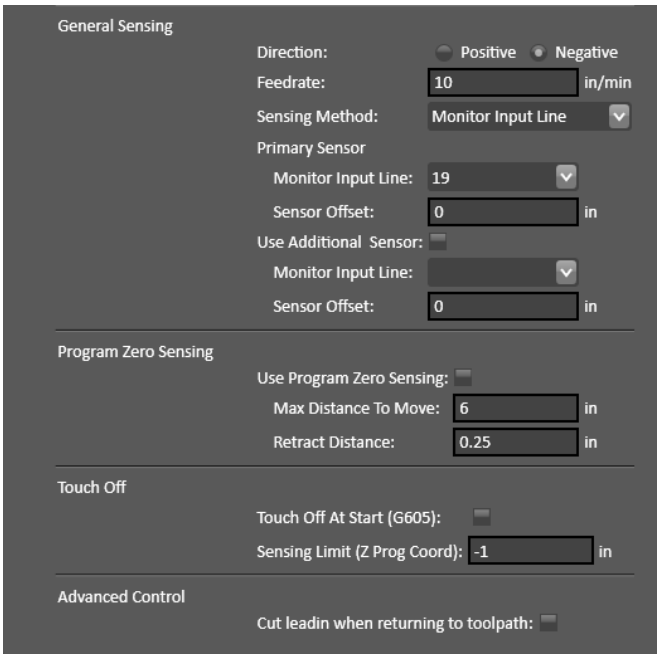
  Waterjet On/Off – Enter the proper M-Codes for turning the waterjet on and off. Typically, these are 40 and 41, respectively. Note that the M-Codes must be defined in the M-Code Definitions.

  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.



- **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.
- **Automatic Execution Mode**

  Start/End of Cut – Enter the Automatic M-Codes or Macros for turning the cutter on for cuts and off for rapid movements. Note that the Macros and M-Codes must be defined in the M-Code Definitions and Execution Screens.



- **General Sensing**

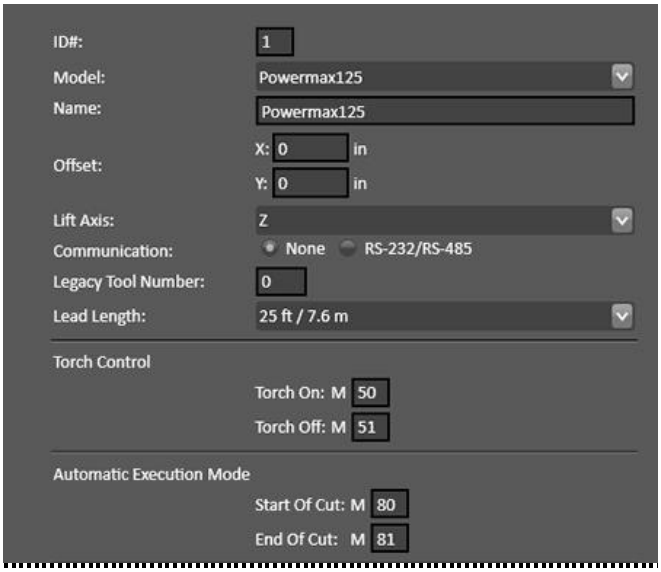
  Direction – Select direction.

  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.

Plasma

When defining a new plasma head, select the **Type**, **Subtype**, and **Model** 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.
- **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.
- **Legacy Tool Number** – Enter number here.
- **Lead Length** – Length of the plasma table.

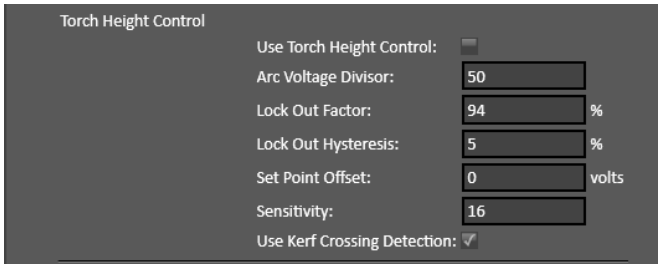
- **Torch Control**

Torch On/Off – Fill in the proper M-Codes for turning the torch on and off. Typically, there are 50 and 51, respectively. Note that the M codes must be defined in the M-Code Definitions.

- **Automatic Execution Mode**

Start/End of Cut – Enter the Automatic M-Codes or Macros for turning the torch and THC on for cuts and off for rapid movements. Note that the Macros and M-Codes must be defined in the M-Code Definitions. Automatic Execution must be enabled in M-Code Execution.

- **Torch Height Control**



Use Torch Height Control – Select this value to enable torch height control, if your torch has it.

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

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

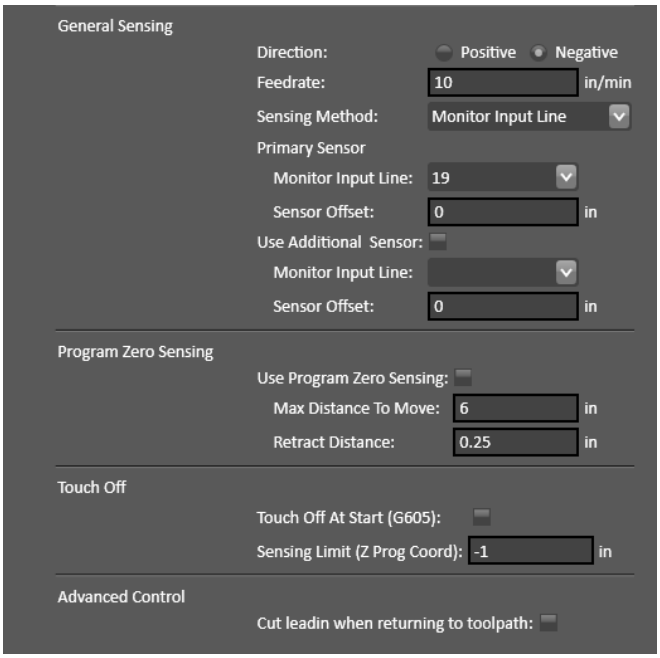
Lock Out Hysteresis - The THC will turn back on again when the XY velocity goes above the ((Lock Out Factor) + (Lock Out Hysteresis))* (Programmed Feedrate).

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

Sensitivity - Determines how sensitive the THC acts towards voltage fluctuation. This number can vary from 1-64 where a 1 is the most sensitive and a 64 is the least sensitive.

Use Kerf Crossing Detection – Turns off THC while crossing a previous cut on the sheet to prevent unwanted diving.

- **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.

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**

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

Sensing Limit – The lowest coordinate that the Z axis will sense to.

- **Advanced Control**

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

Axes

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

Motor Line/Axis Mapping

FlashCut provides three signals for motor drivers: step, direction, and enable. Different driver manufacturers have varying requirements for the polarity and timing of these signals. FlashCut provides the flexibility to tailor the motor signals to run any driver. Motor lines are the physical lines on the signal generator used for driving motors.

Each column is described below:

Line

This number corresponds to the axis number on the back of the CNC controller. Each signal generator can command up to five independent signals simultaneously.

Axis Letter

Assigns a logical axis letter (X, Y, Z, A/W, B/V, C/U) to a physical signal generator motor line (1, 2, 3, 4, 5). If you have more than one motor driving an axis on your machine, you may assign the same axis letter to more than one motor line.

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.

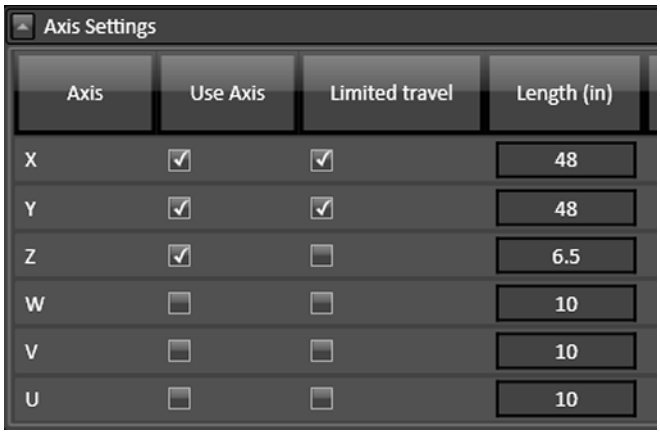
If one homing switch is used for multiple sub axes, or if multiple sub axes are not being used, this sub axis offset value should be zero.

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



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.

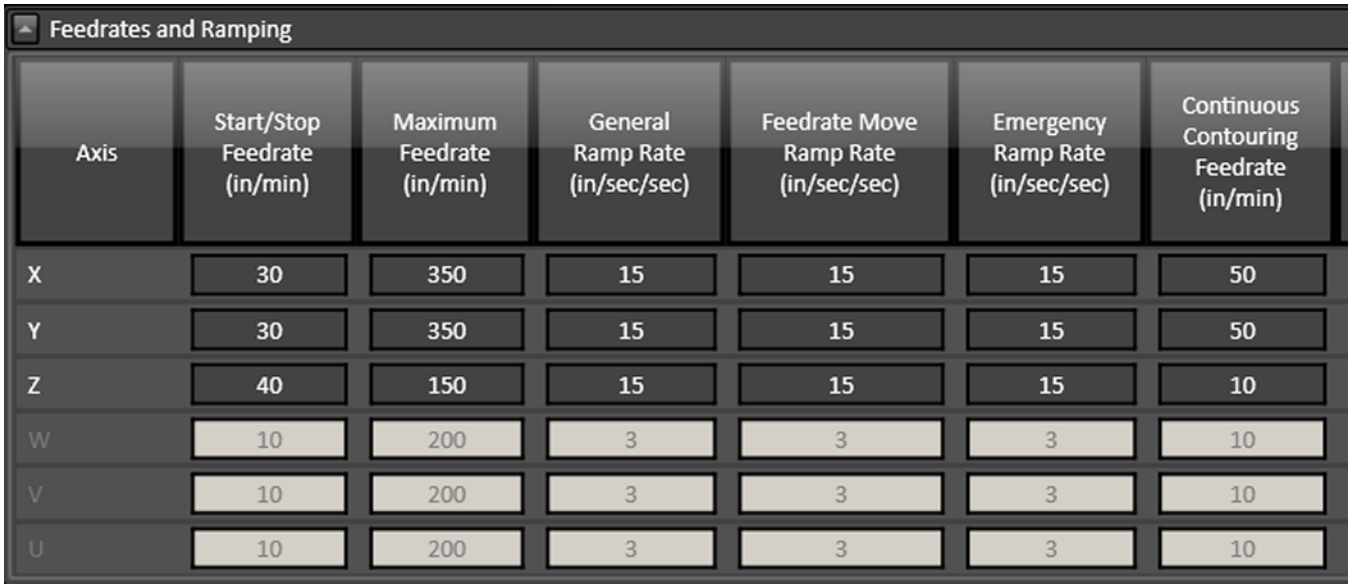
Limited Travel

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

Length (Inches)

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.

Feedrates and Ramping



Axis	Start/Stop Feedrate (in/min)	Maximum Feedrate (in/min)	General Ramp Rate (in/sec/sec)	Feedrate Move Ramp Rate (in/sec/sec)	Emergency Ramp Rate (in/sec/sec)	Continuous Contouring Feedrate (in/min)
X	30	350	15	15	15	50
Y	30	350	15	15	15	50
Z	40	150	15	15	15	10
W	10	200	3	3	3	10
V	10	200	3	3	3	10
U	10	200	3	3	3	10

For your system to run accurately, it's important to tune the values on this panel correctly for your specific combination of drivers, motors and machine tool. All parameters must be set conservatively to prevent the system from losing position (losing steps).

An axis can lose a step when the motor gets the signal to move a step, but isn't physically able to perform the move. The usual cause is insufficient torque at a given motor RPM. Since most stepper motors are open loop systems, there is no way of telling when a step is lost without physically measuring the movement of the axis and comparing that to the amount it should have moved. However, when stepper motors are not over torqued they are very reliable and accurate.

For that reason, we highly recommend finding the values for the following settings at which steps are not lost, then limiting the settings to about 70% of those values. Due to variations in the drive mechanism for each axis, make sure you follow the instructions below at several positions along each axis, in both directions:

Axis

These are the six possible axis identifiers that are expected to be found in a G-Code program. If the G-Code program calls any of these axis letters, the axis must be selected with the **Use Axis** checkbox. There are additional parameters that are specific to each axis.

Start/Stop Feedrate (in/min)

Sets the initial and ending feedrate for ramped moves.

To set the start/stop feedrates:

a. Enter 499 for the X axis Start/Stop Feedrate, and 500 for the Maximum Feedrate, and then choose OK.

If your software settings do not allow you to set the Maximum Feedrate to 500, set it to the largest value the software allows, and set the Start/Stop feedrate to 1 less than that.)

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

c. Choose the Point button in the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Feedrate from the Rate Mode pull-down menu and enter a relatively slow feedrate (such as 5) in the Linear Feedrate text box. Make sure you have room to move the X axis 1 inch, and then choose Move.

d. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.

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

f. Now run the entire length of the X axis in both directions to make sure there is no slippage at any point on the entire axis.

g. Choose Feedrate/Ramping from the Configuration menu.

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

i. Repeat this process for all axes.

Maximum Feedrate (in/min)

Sets the maximum feedrate for the system. No matter how fast the system is commanded to move (from the G-Code, Point, Jog or MDI control panels), the axis will never move faster than this value. The system will use ramping (acceleration and deceleration) to achieve this feedrate.

To set the maximum feedrates:

Make sure you've set the Start/Stop Feedrates correctly as described above.

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

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

c. Choose the Point button on the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Feedrate from the Rate Mode pull-down menu and enter a feedrate that is double the Start/Stop Feedrate for the X axis. Make sure you have room to move the X axis 1 inch, and then choose Move.

d. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.

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

f. Now run the entire length of the X axis in both directions to make sure there is no slippage at any point on the entire axis.

g. Choose Feedrate/Ramping from the Configuration menu.

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

i. Repeat this process for all axes.

General Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for rapid moves and jogging.

Ramping Rates typically range from 1 to 10 in/sec^2. Slower ramping rates require more time to ramp up to the maximum feedrate and to ramp down to a stop. This may become a potentially dangerous situation when using the Feed Hold button or jogging since the machine will take longer to come to a complete stop. The goal is to choose a fast ramping rate that will start and stop the tool responsively without losing steps. Fast ramping rates can also allow acceleration past resonant speeds of a stepper motor.

Make sure you've set the Start/Stop and Maximum Feedrates correctly as described above.

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

b. Choose OK.

c. Choose the Point button on the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Rapid from the Rate Mode pull-down menu. Make sure you have room to move the X axis 1 inch, and then choose Move. If the table doesn't reach a continuous speed before ramping down to a stop, there isn't enough room to fully ramp up and down, and you should increase the length of the move.

d. If the motor slips, repeat this process with a lower ramping rate. If the motor doesn't slip, try a higher ramping rate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.

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

f. Choose Feedrate/Ramping from the Configuration menu.

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

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

i. Repeat this process for all axes.

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

Feedrate Move Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for feedrate moves.

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

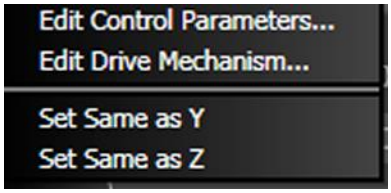
Axis Drive Parameters

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):



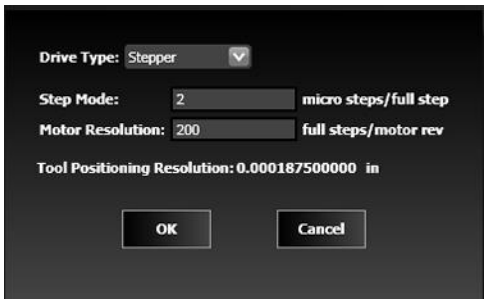
Set Same as

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.

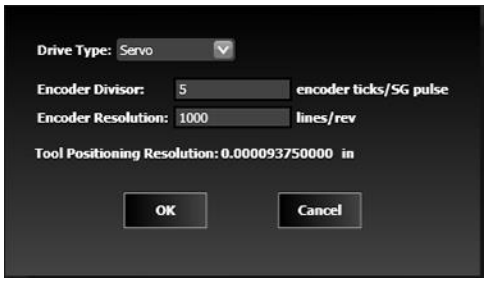
- **Stepper**



Step Mode – The number of micro steps between each full motor step. Enter "1" for full-stepping, "2" for half-stepping, "4" for quarter-stepping, and so on.

Motor Resolution – The number of full motor steps for one revolution of the motor. For example, a 1.8° stepper motor has 200 full steps per revolution; a 0.9° stepper motor has 400 full steps per revolution.

- **Servo**



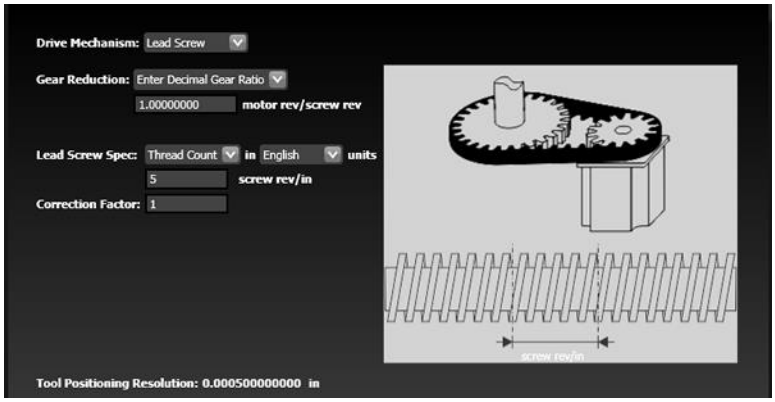
Encoder Divisor – The number of encoder pulses that correspond to one pulse from the Signal Generator.

Encoder Resolution – The number of encoder pulses per revolution, not including any adjustment for running in quadrature mode. When calculating the Tool Positioning Resolution, FlashCut assumes the encoder is running in quadrature mode.

Edit Drive Mechanism...

Opens the Drive Mechanism dialog box. Select a drive mechanism from the Drive Mechanism pull-down menu. Note that Rotary is an option only for axes A, B and C.

- **Lead Screw**

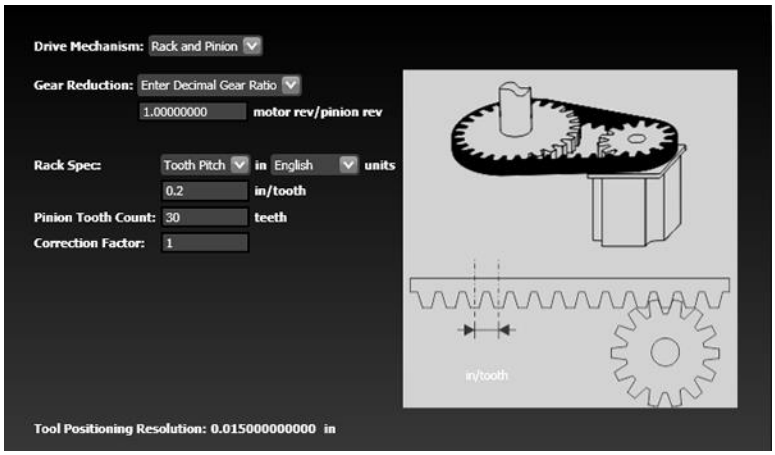


Gear Reduction – Enter the parameters for any gearing between the motor and lead screw. A decimal gear ratio of 1.0 indicates no gearing.

Lead Screw Specification – Enter the parameters for the lead screw.

Correction Factor – Use this value to fine tune the axis, so the distance it moves exactly matches the commanded distance.

- **Rack and Pinion**



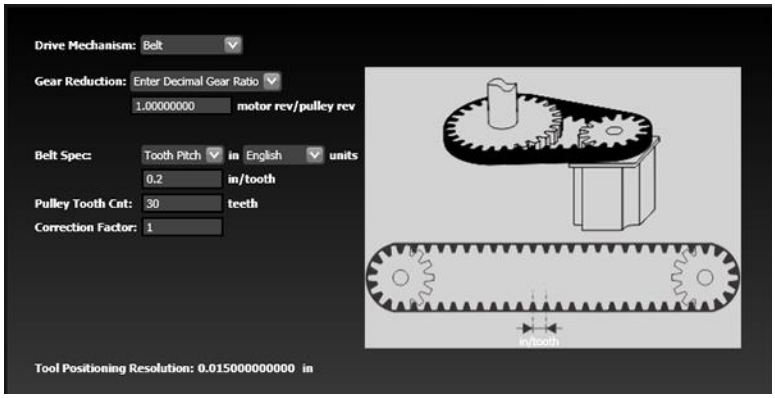
Gear Reduction – Enter the parameters for any gearing between the motor and pinion. A decimal gear ratio of 1.0 indicates no gearing.

Rack Specification – Enter the parameters for the rack.

Pinion Tooth Count – The number of teeth on the pinion.

Correction Factor – Use this value to fine tune the axis, so the distance it moves exactly matches the commanded distance.

- **Belt**



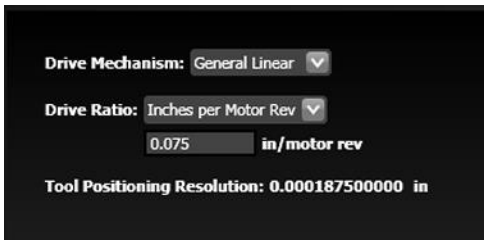
Gear Reduction – Enter the parameters for any gearing between the motor and drive pulley. A decimal gear ratio of 1.0 indicates no gearing.

Belt Specification – Enter the parameters for the belt.

Pulley Tooth Count – The number of teeth on each pulley.

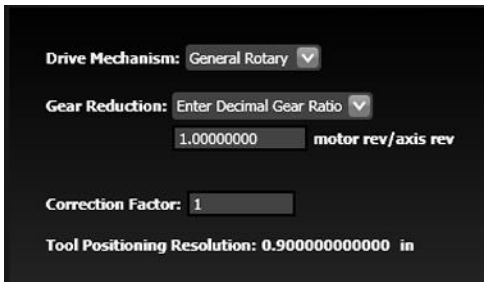
Correction Factor – Use this value to fine tune the axis, so the distance it moves exactly matches the commanded distance.

- **General Linear**



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

- **Rotary**



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

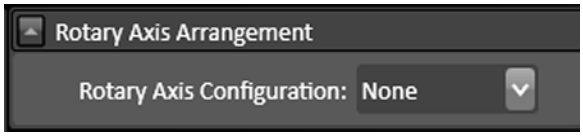
Correction Factor – Use this value to fine tune the axis, so the distance it moves exactly matches the commanded distance.

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

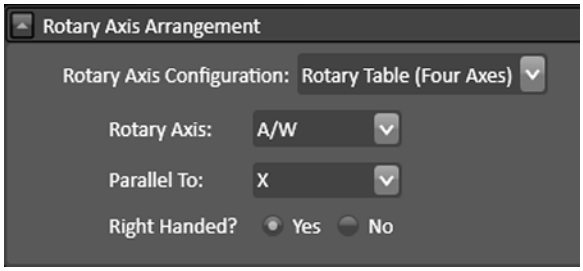
Select **None** if you are not using a rotary axis.

Other

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

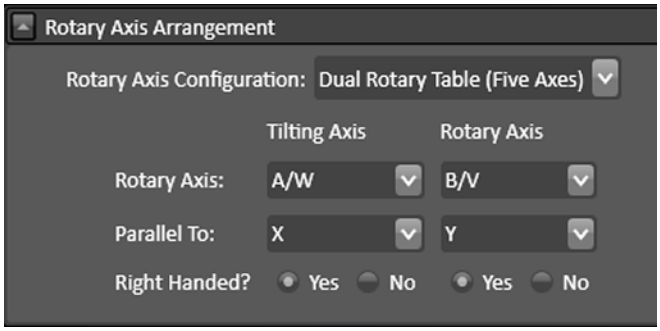
Rotary Table (Four Axes)

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.



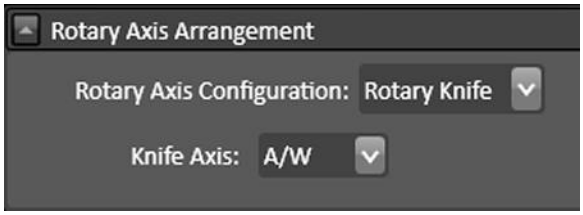
Dual Rotary Table (Five Axes)

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



Rotary Knife

Select Rotary Knife if you are running a CNC knife cutting machine. You must also select a Knife Axis (A, B, or C).



General

These options are part of the rotary table configuration screens.

- Use offset for rotary zero

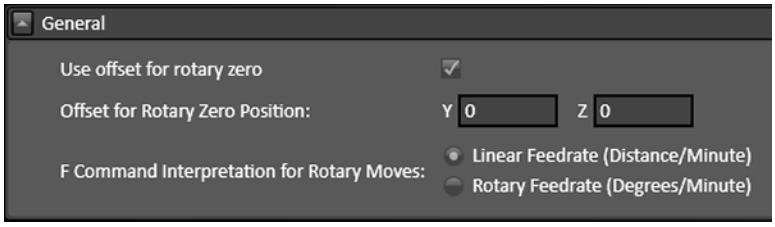
  If checked, define the offset below.

- Offset for Rotary Zero Position

  Indicates the fixture offset for the rotary zero position.

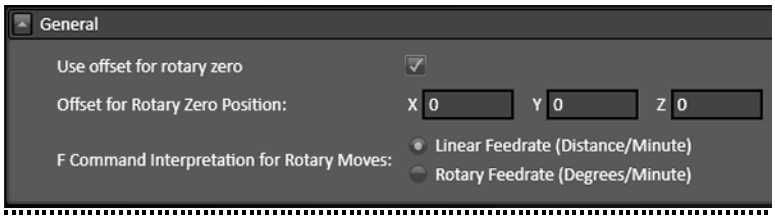
  - 4-Axis machining

    Rotary zero is anywhere along the axis of rotation of the rotary axis. The parallel axis in the fixture offset is ignored.



  - 5-axis machining

    Rotary zero is at the intersection of the axes of rotation of both rotary axes.



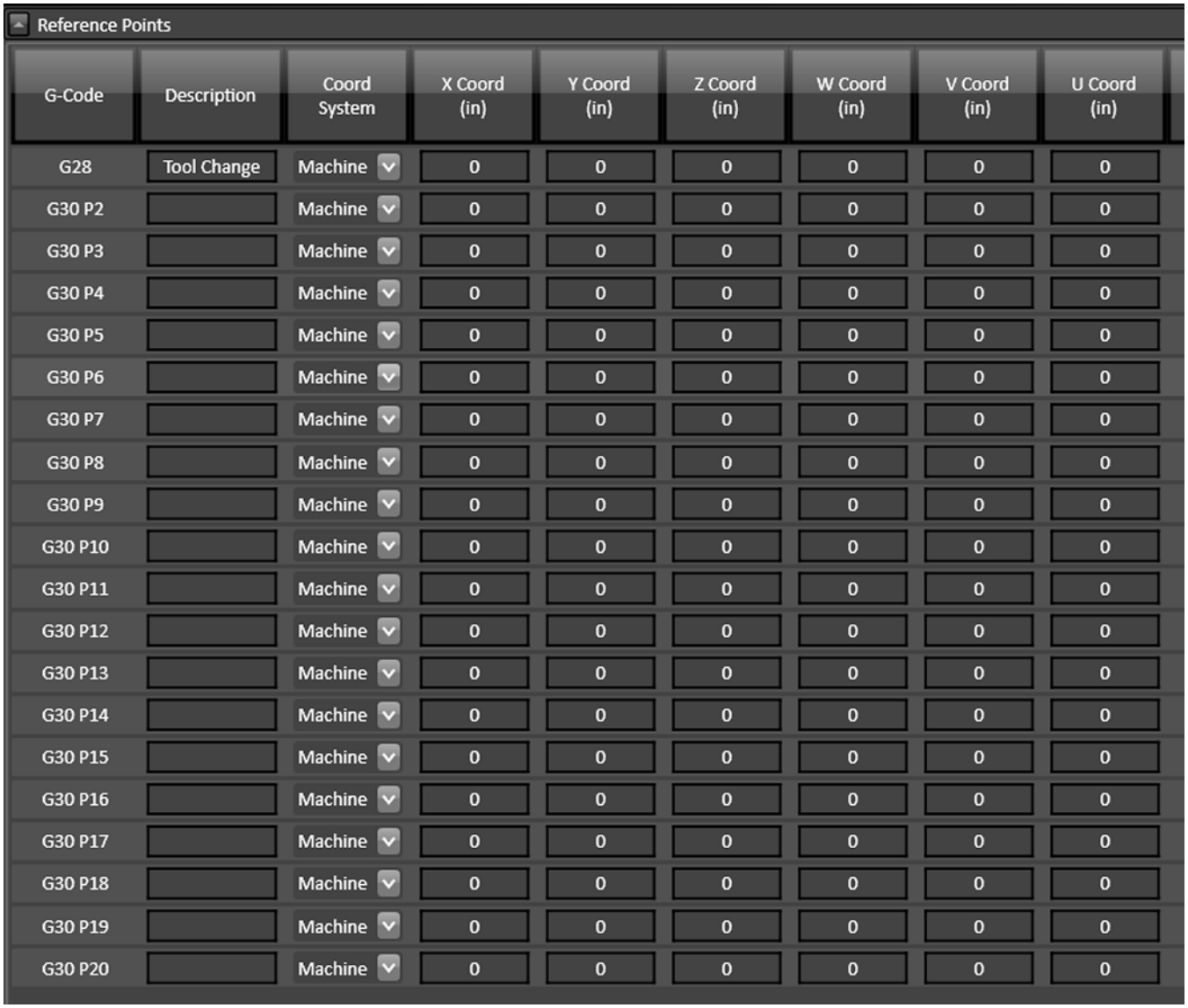
- F Command Interpretation for Rotary Moves

  For moves that include rotary motion, this setting tells FlashCut whether to interpret the feedrate command (F) as a linear (distance/minute) or rotary (degrees/minute) feedrate.

Reference Points

Reference Points

Reference points provide a way to move the machine tool to a predefined location. They are accessible using the G28 and G30 commands, described in the Programming Reference.



G-Code	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.

Homing

Axis Settings

For each axis, the following settings are available:

Home Axis

Select to configure homing for the axis.

Home End

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

Home Order

The axis order for the homing operation. For safety, set the order so the initial homing move retracts the cutting tool away from the workpiece.

Home Switch Offset

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

Homing Rate

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

Homing Tolerance

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

Execute M-Codes Before / After

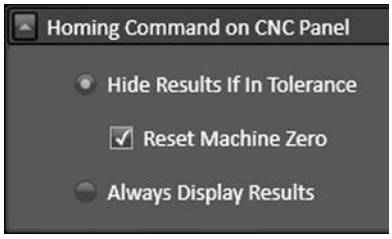
You may specify M-Codes to execute before and after each axis homes. For more information on defining M-Codes see **M-Code Definitions**.

Home Position Machine Coordinate

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

Homing 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

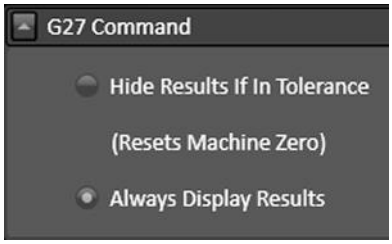
Tells FlashCut to automatically reset Machine Zero to the new location found, if the discrepancy from the previous Machine Zero location is within the Homing Tolerance for all axes.

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.

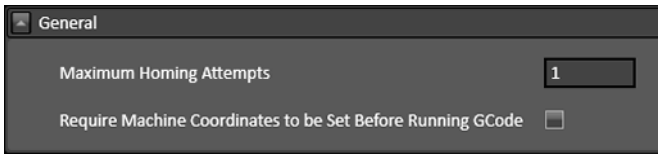
Always Display Results

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 GCode

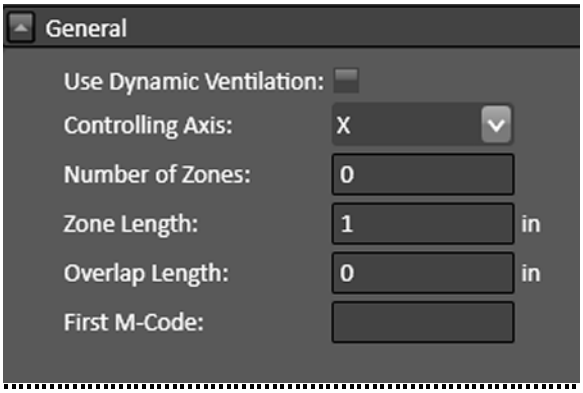
When checked, machine coordinates must be set before the system can run a G-Code program.

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.

Number of Zones

Sets the number of zones.

Zone Length

Sets the length of each zone along the controlling axis.

Overlap Length

Sets the amount of overlap between zones. Overlapping zones ensure that the dampers controlling the airflow have sufficient time to open and close.

First M-Code

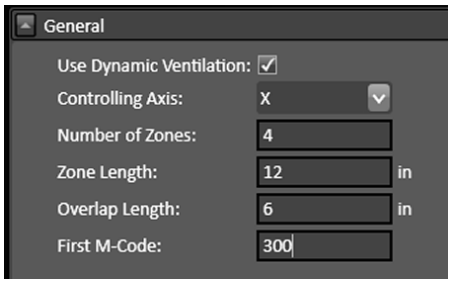
Enter the first M-Code here. The first M-Code is the code that opens the damper in zone one. Note that **codes are consecutive**: the second M-Code will close the damper for zone one, and so on.

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

Dynamic Ventilation Example

The following steps illustrate how a dynamic ventilation system for a plasma table could be configured in FlashCut. The example assumes a table with four 1-foot zones along the X axis, with a recommended overlap of 6 inches. **The values chosen are arbitrary, and are only an example**. Consult the manufacturer of your system to determine the optimal values for your ventilation system.

1. Enter the following values in the Dynamic Ventilation General configuration panel:



2. Define Output Line M-Codes 300, 301, 302 and 303 to control the 4 dampers in each zone on the X axis (see M-Code Definitions, below). It is important to use consecutive numbers for the codes, and to begin with the zone starting at the X axis zero point.

3. Save your setup file.

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

Again, consult the manufacturer of your plasma table and ventilation system to determine the optimal values for configuring your system.

Programming

- M-Code Definitions
- M-Code Execution

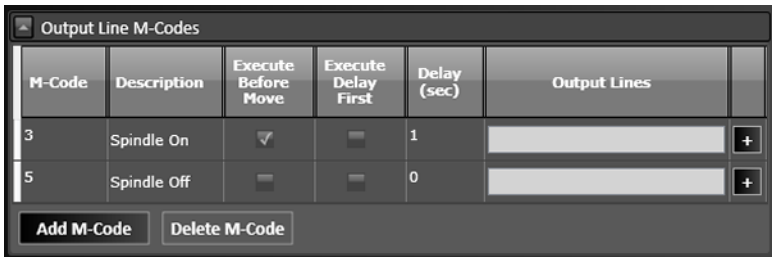
M-Code Definitions

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.

Description

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

Execute Before Move

Indicates the sequence of actions when there is a machine tool move command on the same program line as the M-Code. Check this box to execute the M-Code before the machine moves as commanded.

Execute Delay First

Check this box to implement the specified delay before the M-Code. If the box remains unchecked the delay will occur immediately after the M-Code.

Delay (sec)

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

Output Lines

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

Add M-Code Button

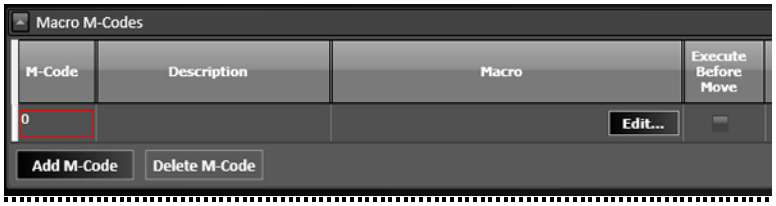
Adds a new line to define an M-Code.

Delete M-Code Button

Deletes the highlighted M-Code definition.

Macro M-Codes

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



M-Code

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

Description

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

Macro

Click the **Edit** button to bring up the G-Code 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

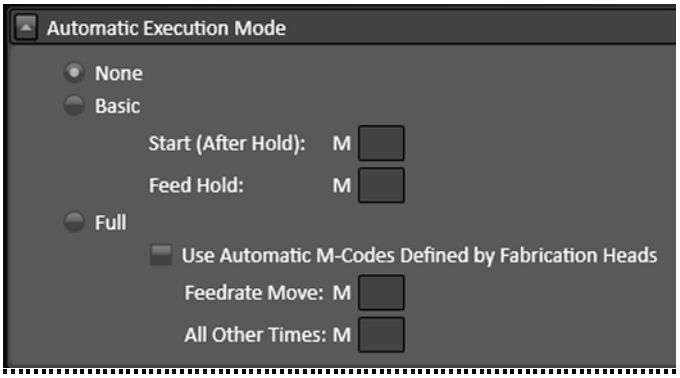
The M-Code execution option presents these M-Code execution configuration types:

- Automatic Execution Mode
- 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.

Automatic Execution Mode

Enter an M-Code in the field(s) associated with the selected radio button.

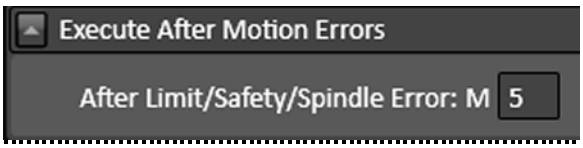


- **None** – FlashCut doesn't execute any M-Codes automatically.
- **Basic** – This mode is intended for applications where it's convenient to automatically turn off spindle, vacuum and other devices whenever the machine is stopped.
  - **Start (After Hold)** FlashCut can execute an M-Code whenever **Start** is clicked after a feed hold has occurred (but not when initially starting the file).
  - **Feed Hold** FlashCut can execute an M-Code whenever **Feed Hold** is clicked, or an error occurs.

Note You don't need to fill in both M-Code fields. For example, you can specify a **Feed Hold** M-Code, and leave the **Start (After Hold)** field blank.

- **Full** – This mode is intended for applications where the signal generator must turn output lines on during feedrate moves and off during rapid moves. Applications include plasma cutting, laser cutting, waterjet cutting, and liquid dispensing.
  - **Feedrate Move** FlashCut executes the **Feedrate Move** M-Code before any feedrate move (G01, G02, G03), and before restarting a feedrate move after a Feed Hold.
  - **All Other Times** FlashCut executes the **All Other Times** M-Code:
    - Before any rapid move (G00)
    - Whenever **Feed Hold** is hit during a feedrate move
    - At the end of the program if the program ends with a feedrate move
    - When an error occurs

Execute After Motion Errors



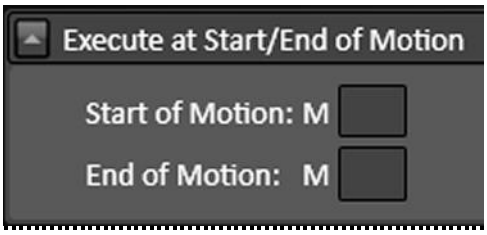
- **After Limit/Safety/Spindle Error** – FlashCut executes the M-Code after an error occurs. This is useful in applications where it's safest to automatically turn off the spindle, vacuum and other devices whenever the signal generator detects an error.

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.

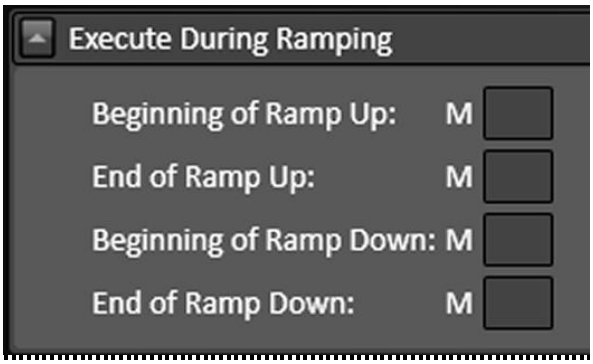


- **Start of Motion** – FlashCut executes this M-Code immediately before any motion begins (including starting a G-Code program, jogging, and so on).
- **End of Motion** – FlashCut executes this M-Code immediately after motion stops.

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.
- **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.

I/O

- Input Lines

Input Lines

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

General

Sets the duration in sections of the debounce for operations M100 and M101. This is the duration that a switch must be continuously tripped for FlashCut to recognize it as tripped.

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

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.
- **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.
- **Load File** – FlashCut will load the specified G-Code file.
- **Unused** – The input line is not used and will be ignored by the system.

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.

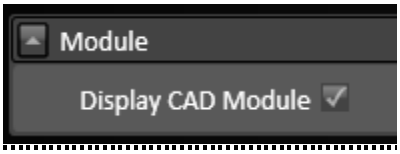
CAD

- General
- Features

General

Display CAD Module

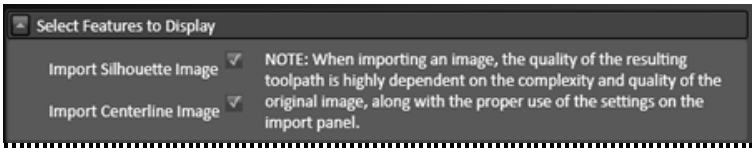
When selected, the CAD module will be available in FlashCut.



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.

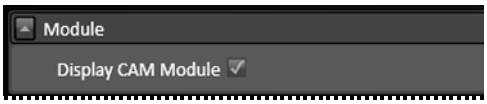
CAM

- General
- Features
- Breaks
- Post Processor
- Toolpath Adjustments

General

Module

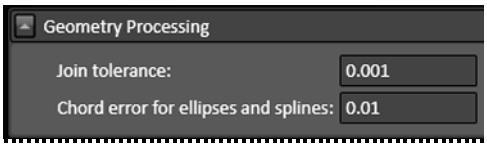
When selected the CAM module will be available in FlashCut.



Geometry Processing

Join tolerance

Defines tolerance in system units



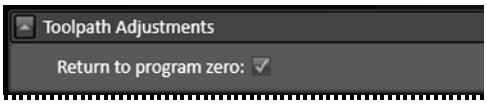
Chord error

Defines amount that will trigger an error in system units.

Toolpath Adjustments

Return to program zero

When selected, this feature is enabled.

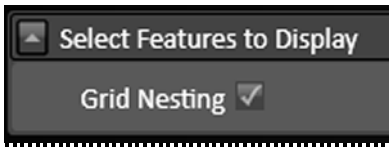


Features

The Features configuration option presents the following configuration items:

Select Features to Display

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



Grid Nesting

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

Breaks

Breaks on Parts

This section defines the default values for the break placements and lead definitions **for parts**. The parameters are identical to the section below titled **Breaks on Cutouts**, so it is important to distinguish between the two. A part is a closed geometric shape that requires the toolpath to be offset to the outside. There is a lead in and lead out definition for two break placement approaches. The break can be placed at a corner or not at a corner.

Enable starting at corners

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.

- Minimum corner angle
- Maximum corner angle

Starting at corner

If an acceptable angle is found, a break will be placed at the vertex of the angle. The break will then use the defined lead in and lead out values below.

- Overburn

  To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.

- Lead in

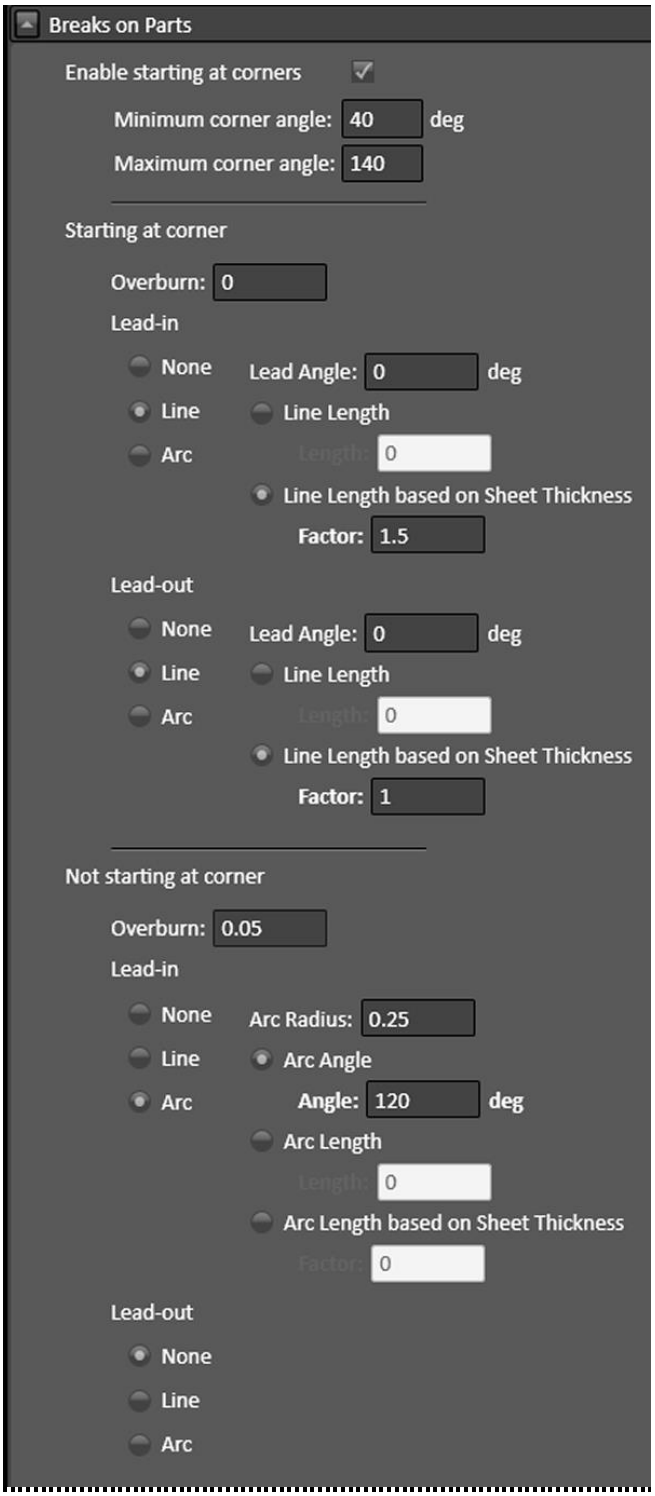
  You may choose from:

  - None
  - Line
  - Arc

- Lead out

  You may choose from:

  - None
  - Line
  - Arc



Not starting at corner

When no acceptable angle is found, the break will not be placed at a corner. It will instead be placed at the midpoint of the longest line. If no lines exist the break will be placed on the arc nearest the origin. Once the break is placed on the toolpath, the lead in and lead out will be created based on the definitions below.

- Overburn

  To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.

- Lead in

  You may choose from:

  - None
  - Line
  - Arc

- Lead out

  You may choose from:

  - None
  - Line
  - Arc

Breaks on Cutouts

This section defines the default values for the break placements and lead definitions **for cutouts**. This is identical to the **Breaks on Parts** section above, so it is important to distinguish between the two. A cutout is any geometric shape inside a part. Cutouts require the toolpath to be offset to the inside. There is a lead in and lead out definition for two break placement approaches. The break can be placed at a corner or not at a corner.

Enable starting at corners

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.

- Minimum corner angle
- Maximum corner angle

Starting at corner

If an acceptable angle is found, a break will be placed at the vertex of the angle. The break will then use the defined lead in and lead out values below.

- Overburn

  To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.

- Lead in

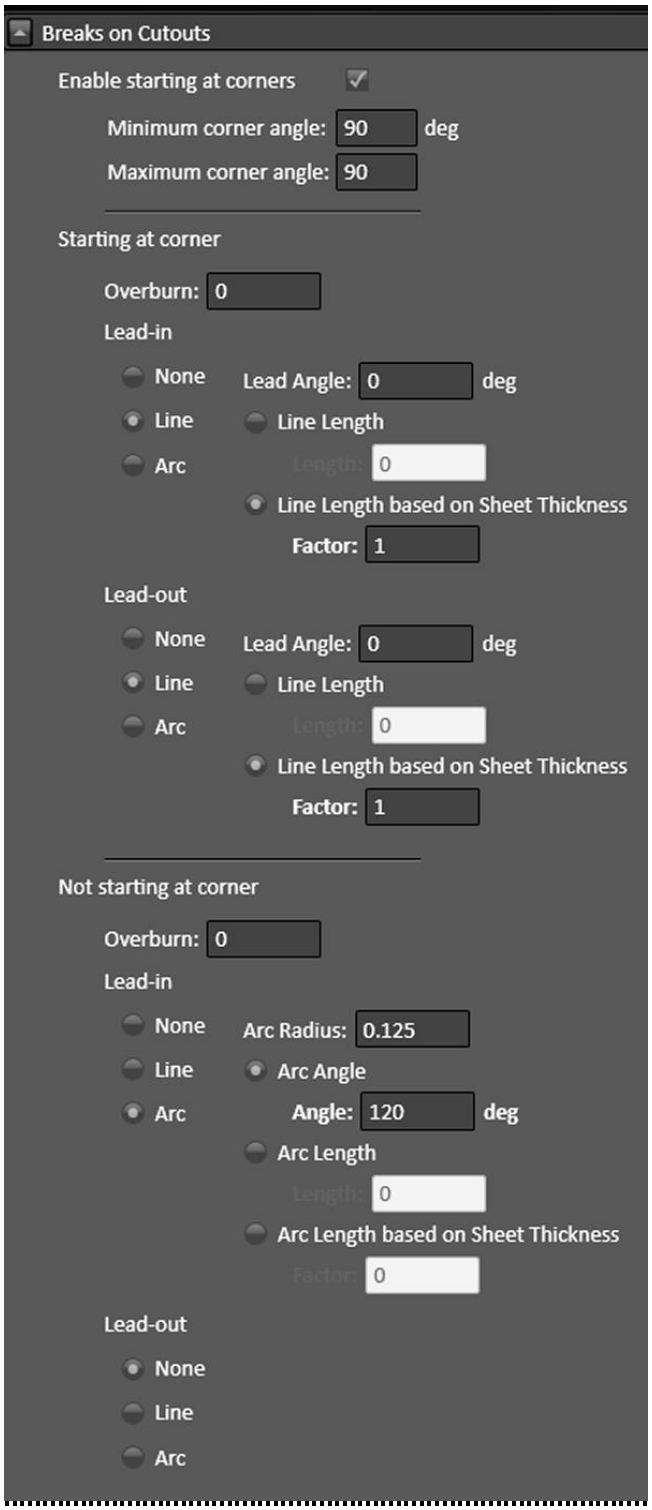
  You may choose from:

  - None
  - Line
  - Arc

- Lead out

  You may choose from:

  - None
  - Line
  - Arc



Not starting at corner

When no acceptable angle is found, the break will not be placed at a corner. It will instead be placed at the midpoint of the longest line. If no lines exist the break will be placed on the arc nearest the origin. Once the break is placed on the toolpath, the lead in and lead out will be created based on the definitions below.

- Overburn

  To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.

- Lead in

  You may choose from:

  - None
  - Line
  - Arc

- Lead out

  You may choose from:

  - None
  - Line
  - Arc

Post Processor

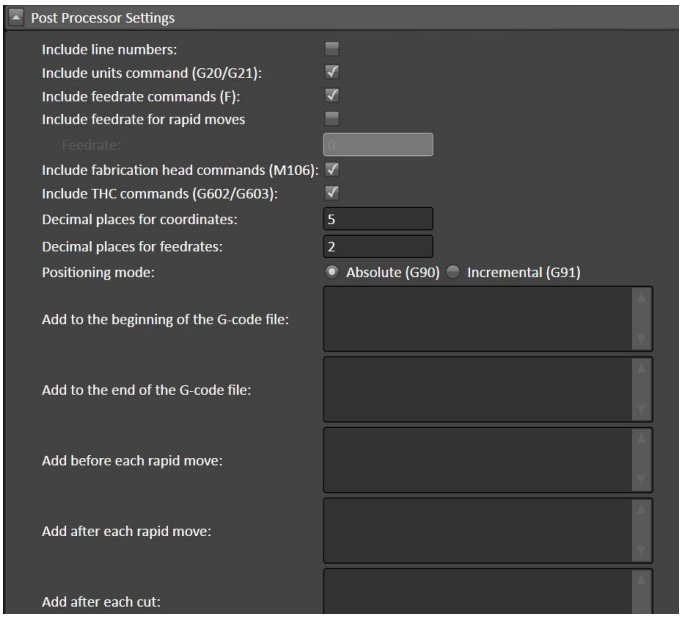
Post Processor Settings

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.

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

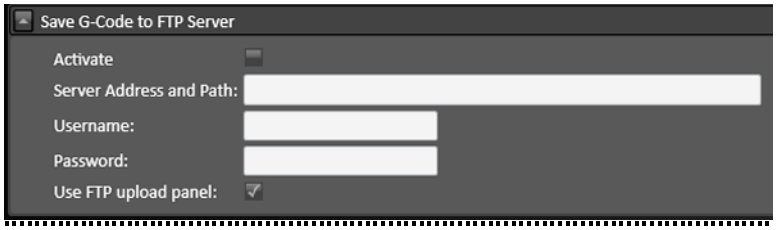
Any G-Code command entered here will be added **after** all rapid moves (G00).

Add after each cut

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

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.

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.

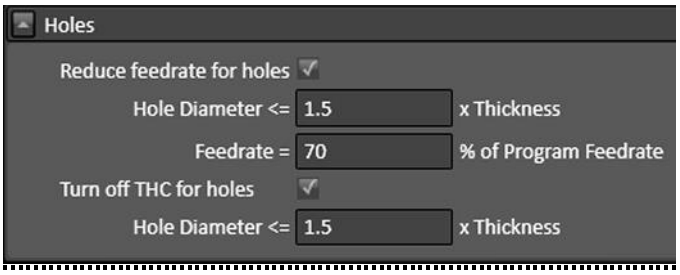
If **Use FTP upload panel** is unchecked: the file is uploaded with a default filename (e.g., gcode.nc).

Toolpath Adjustments

Holes

Reduce feedrate for holes

When checked, feedrate will be reduced for holes of the listed diameter multiplied by the thickness of the material, at the listed percentage of the program feedrate.



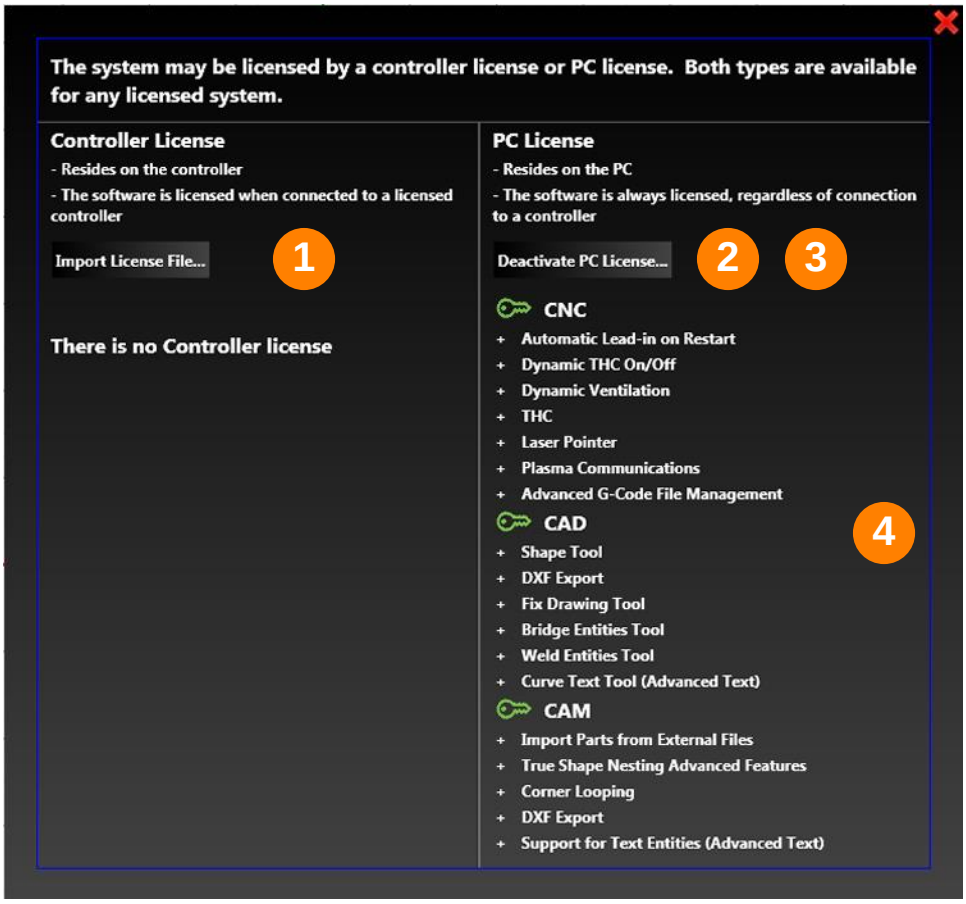
Turn off THC for holes

When checked, torch height control will be turned off for holes of the listed diameter x the thickness multiplied by the thickness of the material.

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.



1. Import License File
2. Activate PC License
3. Deactivate PC License
4. Licensed Features List

License Management Window

Import License File

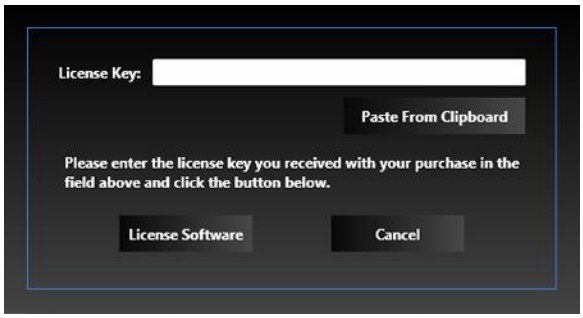
If you've received a license file, you may import the file into this software to license it. Follow these steps to import your license into your software.

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.

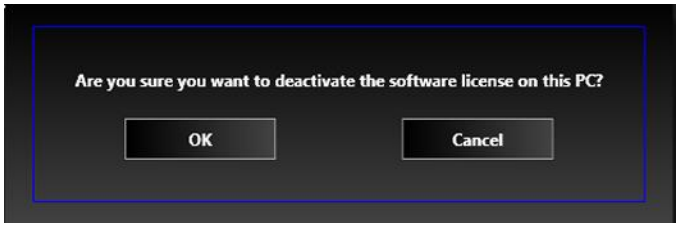


3. If you've copied the license key from the email you received, click the **Paste From Clipboard** button. FlashCut pastes the license key in the **License Key** field. Otherwise, you may type the license key in the field yourself.
4. Click the **License Software** button.
5. FlashCut installs the license 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.

1. Click the **Deactivate PC License...** button.
2. FlashCut displays a confirmation dialog box.



3. Click **OK** to deactivate the license.

Licensed Features List

This list shows which advanced or a la carte features of FlashCut are enabled by the installed license key. For an overview of FlashCut features, see **Program packages and features**.

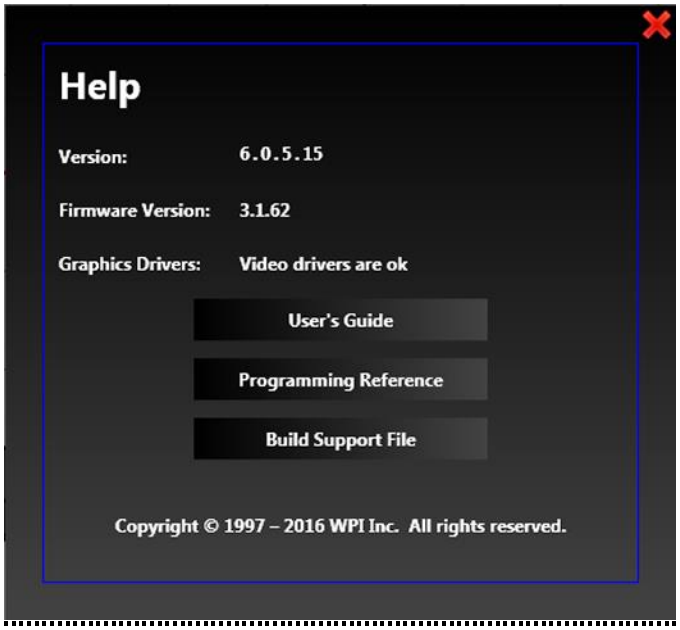
Getting Help

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
- Graphics Drivers
- User's Guide
- Programming Reference
- 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.

Graphics Drivers

The status of the PC's graphics drivers.

User's Guide

Displays this User's Guide in PDF format.

Note Requires a PDF viewer, such as Adobe® Acrobat®

Programming Reference

Displays the FlashCut Programming Reference in PDF format.

Note 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 CADCAM files. If you are having a CADCAM issue, please e-mail the CADCAM 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**.