

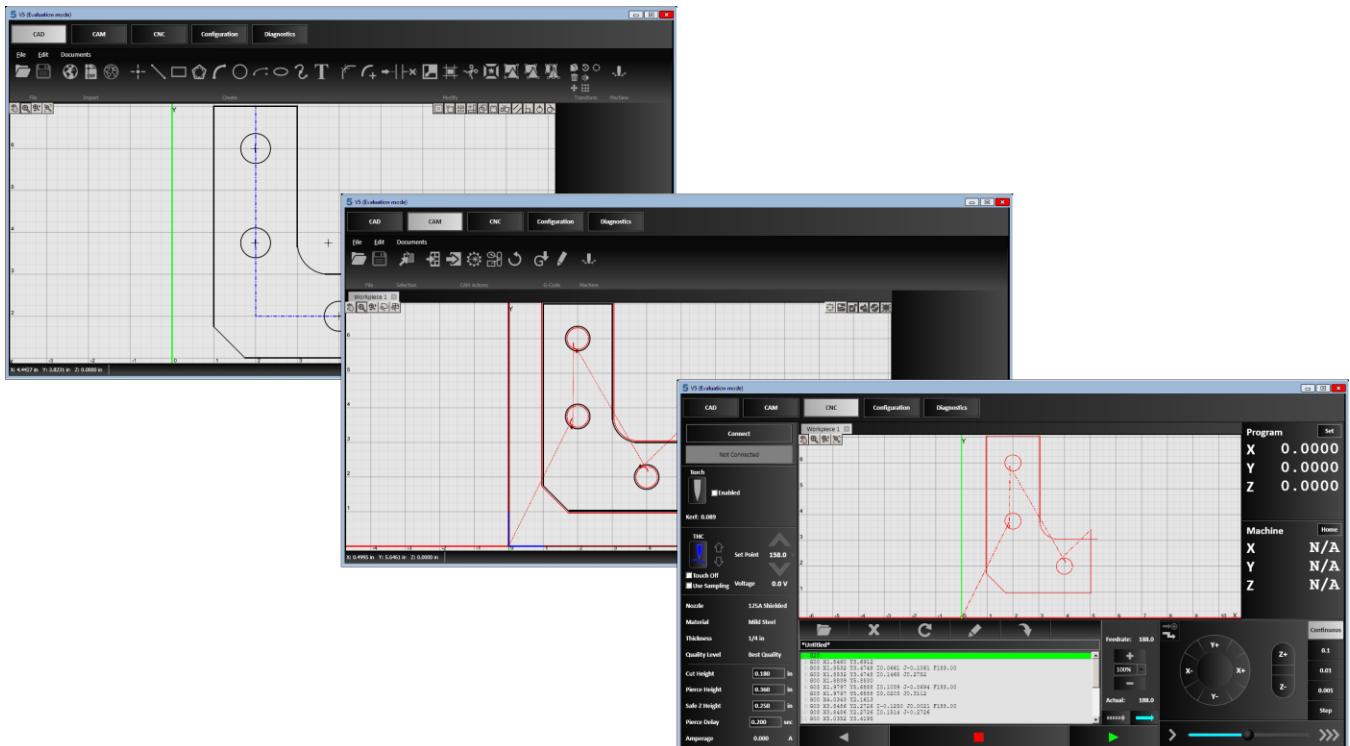
FLASHCUT CNC

CONTROL MADE SIMPLE



FlashCut CAD/CAM and CNC Control Software

Version 5 User's Guide



Notices

Revised May 20, 2015

© 1997-2015 WPI, Inc., all rights reserved

FlashCut CNC® is a registered trademark of WPI, Inc.

Stingray® is a registered trademark of WPI, Inc.

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

When running an automated machine tool, safety is of utmost importance. For proper and safe use of the FlashCut CNC program and your CNC machine, 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 on alert for computer crashes at all times.

About this document

Certain of the screen captures in this document may differ in minor ways from the actual FlashCut application. Those differences are used to make sure what you see is clear and help you understand the descriptions.

Technical support

We provide expert technical support for all of our products. We have many resources dedicated to helping you resolve your problems quickly. Please use these resources in the following order:

Website

<http://www.flashcutcnc.com/>

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

Dealer support

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

Email

support@flashcutcnc.com

Email is the most organized way to convey your issues to our support staff. In your e-mail, please state your problem completely. Include your FlashCut version, the processor and speed of your computer, your version of Windows and your signal generator serial number. Attach your Setup and Tooling files (usually found in a folder named **c:\flashcut data**) and, when appropriate, the G-code file with which you are having problems. Alternatively, you can attach a single FlashCut support file generated by the **Build Support File** command on the **Diagnostics** tab. The support file is in ZIP format and contains all relevant files needed by technical support to resolve your issue. Please see **Build Support File** 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, CST, M-F)

Fax: (847) 940-9315

Contents

Getting started	1
System requirements	1
Installing the FlashCut CNC software.....	2
Installing the USB driver.....	6
Windows 7	7
Windows 8	11
FlashCut CAD.....	12
Tabs.....	13
Menu bar.....	13
File menu	13
Edit menu.....	14
Documents menu	16
Ribbon	17
File tools	17
Import tools.....	18
Create tools	22
Modify tools.....	24
Transform tools.....	26
Machine tool (send drawing to CNC)	28
Pan and zoom tools	28
Pan	28
Zoom Extent	28
Zoom.....	28
Zoom to Fit	28
Snap tools.....	29
Drawing workspace.....	30
Parameters area	30
Status bar	31
FlashCut CAM	32
Tabs.....	33
Menu bar.....	33
File menu	33
Edit menu.....	35
Documents menu	37
Ribbon	37
File tools	37
Selection.....	38
G-code.....	38
CAM Actions	39
Machine (send drawing to CNC)	43
Pan and zoom tools	43
Pan	43
Zoom Workpiece Extents	43
Zoom.....	43
Zoom Toolpath Extents.....	43

Zoom Window	43
Display options tools	44
Cutting directions	44
Kerf	44
Material	44
Parts	44
Rapid moves	45
Tool paths	45
Drawing workspace	45
Feature manipulation	45
Lead in/out settings	46
Program zero	47
Parameters area	47
Status bar	47
FlashCut CNC	48
Tabs	49
Signal generator status	49
Plasma torch settings	50
Program coordinates panel	51
Machine coordinates panel	51
Jog control panel	52
Jog-to-point	52
Manual jogging	53
G-code window	54
Open G-code file	55
Close G-code file	55
Reset G-code	55
Edit G-code	55
Jump to Line	55
File name	55
G-code workspace	55
Feedrate override controls	55
Step/Continuous mode toggle	56
Run in reverse	56
Feed hold	56
Run G-code	56
Pan and zoom tools	57
Pan	57
Zoom Extent	57
Zoom	57
Zoom to Fit	57
Drawing workspace	58
Configuration	59
Tabs	60
Setup file (*.STP extension)	60
Tooling file (*.TLG extension)	61
Menu section	62
System	63
General	63

Controller Settings	64
Preferences.....	67
Languages	68
Jogging.....	68
Display	69
Machine	69
Axes	70
Rotary Axes	75
Sensing.....	76
Program Zero Sensing.....	76
Torch Height Control.....	77
Programming	80
M-code Definitions	80
M-code Execution.....	81
I/O.....	84
General.....	84
Line Setup	85
CAD	87
Features	87
CAM.....	87
Features	88
Breaks	88
Post Processor	91
Configuration workspace.....	92
Diagnostics	93
Tabs.....	94
Version.....	94
Firmware version.....	94
Build Support File button.....	94
Input Lines display	94
Import License File... button	95
Activate PC License... button.....	95
Deactivate PC License... button.....	96

Getting started

Thank you for purchasing FlashCut® CNC, the most innovative, powerful CNC control system for Windows 7 and 8. 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 CNC is a unique Windows application, so you'll need some instruction to get started. Since 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.

It is assumed that you already have a working knowledge of the PC and Windows. If you are not familiar with either of these, please review your PC or Windows user's guides before you use FlashCut CNC.

FlashCut Version 5 has three main portions – FlashCut CAD, FlashCut CAM and FlashCut CNC. All three are completely integrated for seamless operation from drawing a part to creating a toolpath to cutting the part.

- **FlashCut CAD** – (New for Version 5) A unique CAD program for creating and importing drawings. FlashCut CAD together with FlashCut CAM (FlashCut CAD/CAM) is sold as a combined product separate from FlashCut CNC but all three together create one seamless program.
- **FlashCut CAM** – (New for Version 5) A unique CAM program for creating and editing toolpaths and G-code. It can be set up to instantly create toolpaths complete with kerf compensation, lead-ins and lead-outs. FlashCut CAM together with FlashCut CAD (FlashCut CAD/CAM) is sold as a combined product separate from FlashCut CNC but all three together create one seamless program.
- **FlashCut CNC** – The latest version of our control software which can be configured to operate almost any type of automated machine tool. It receives its motion commands from G-code programs. G-code can be created in FlashCut CAD/CAM or from an external CAM program, or it can import a DXF CAD drawing. FlashCut CNC is sold separately from FlashCut CAD/CAM but all three together create one seamless program.

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

System requirements

- Windows Version 7 or 8 (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

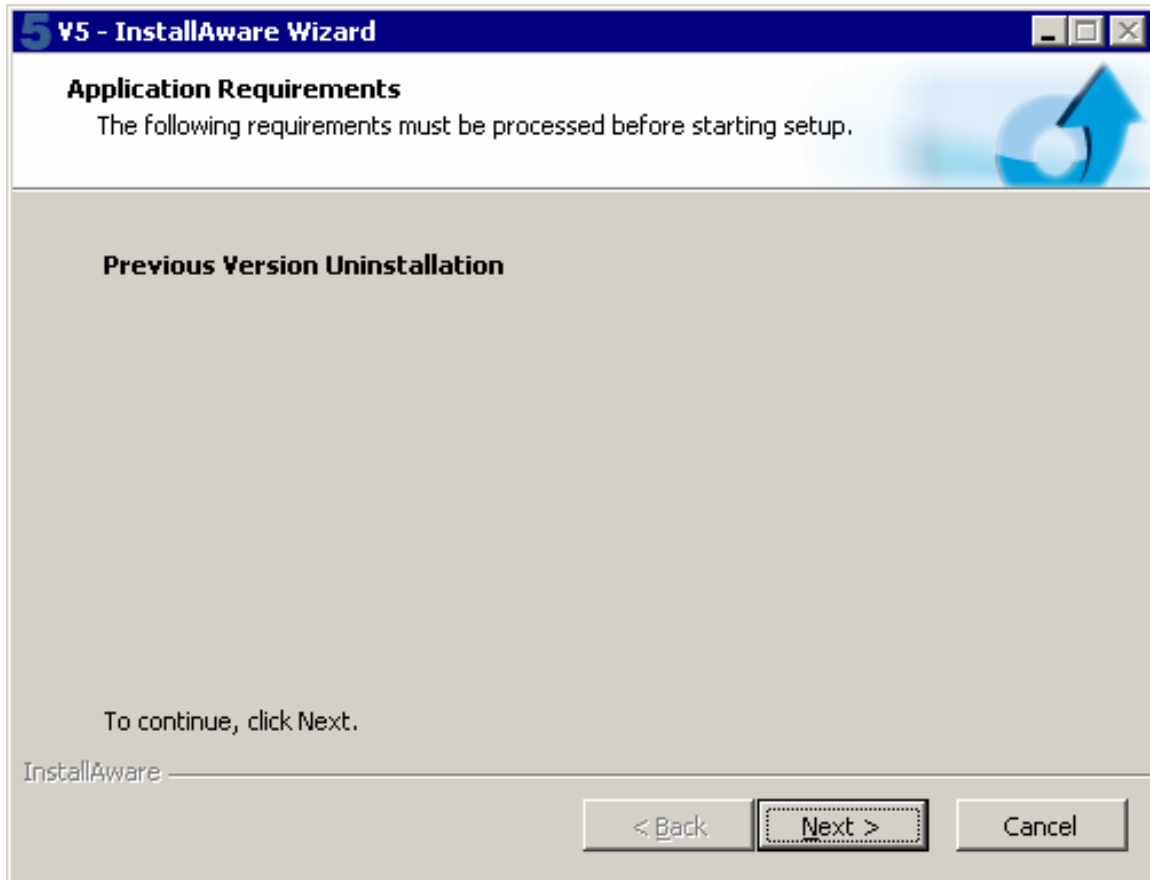
Installing the FlashCut CNC software

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.

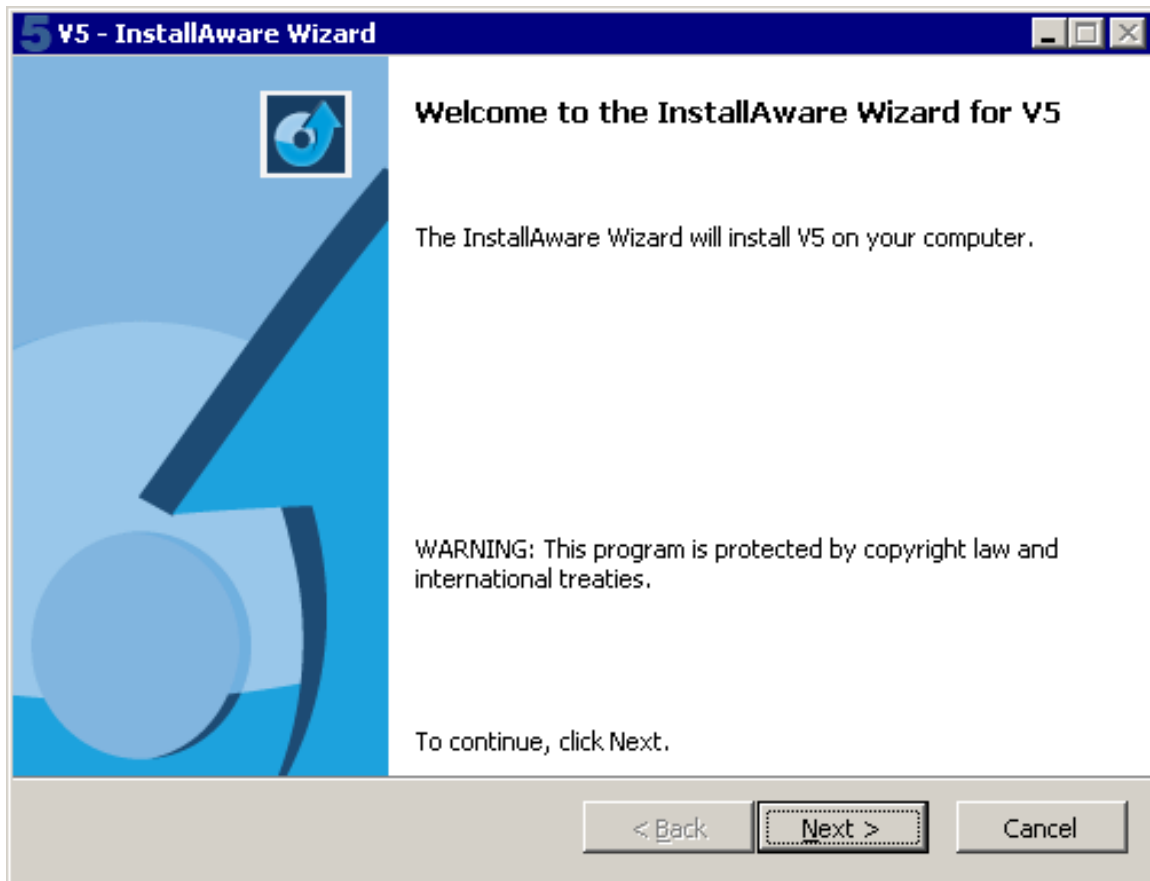
In order to use the FlashCut CNC software, it must first be installed onto the computer. Please follow these instructions.

1. Insert the FlashCut CNC installation CD into the drive or open the installer downloaded from the internet.
2. When prompted to run the installer, select **Run** to open the InstallAware wizard.
3. The wizard first checks for previous versions of FlashCut CNC and prompts you to uninstall them.



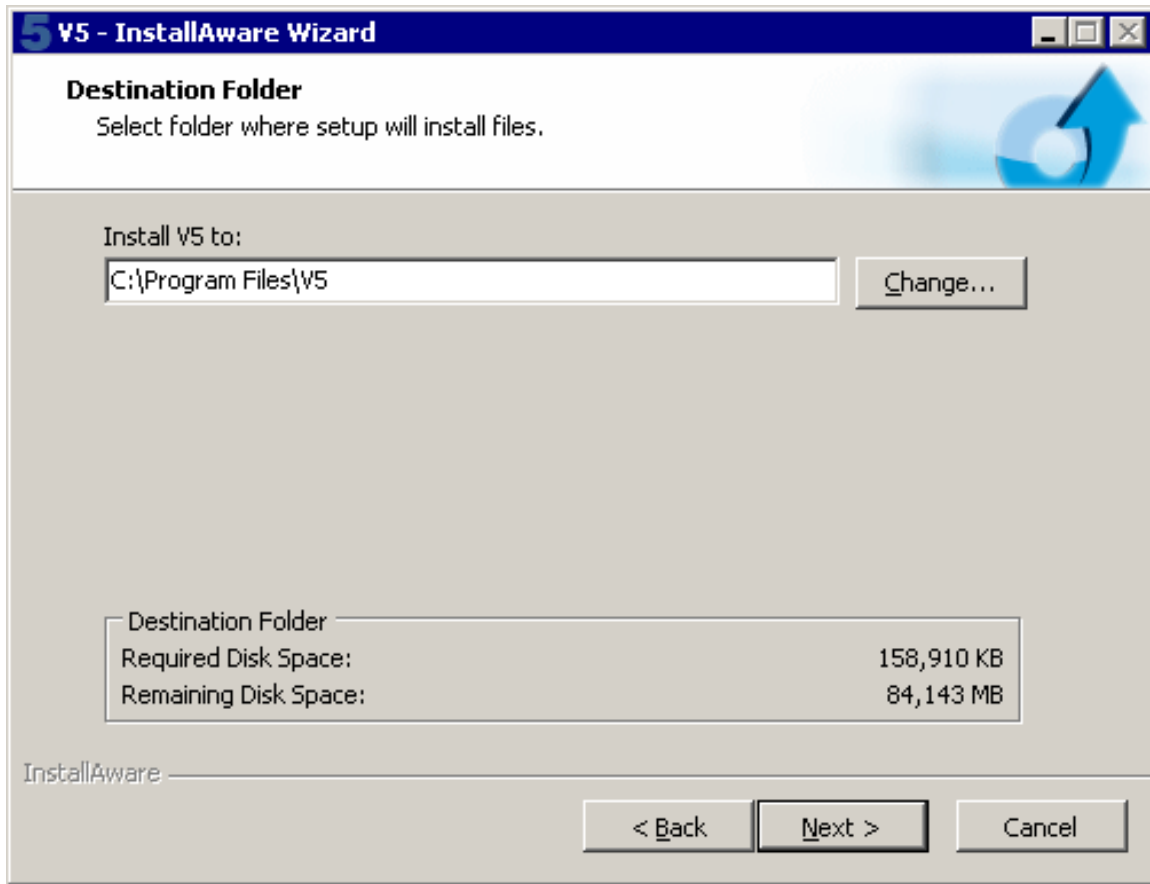
Click **Next >**.

4. After dealing with the preinstallation requirements, the wizard prompts you to install the latest release.



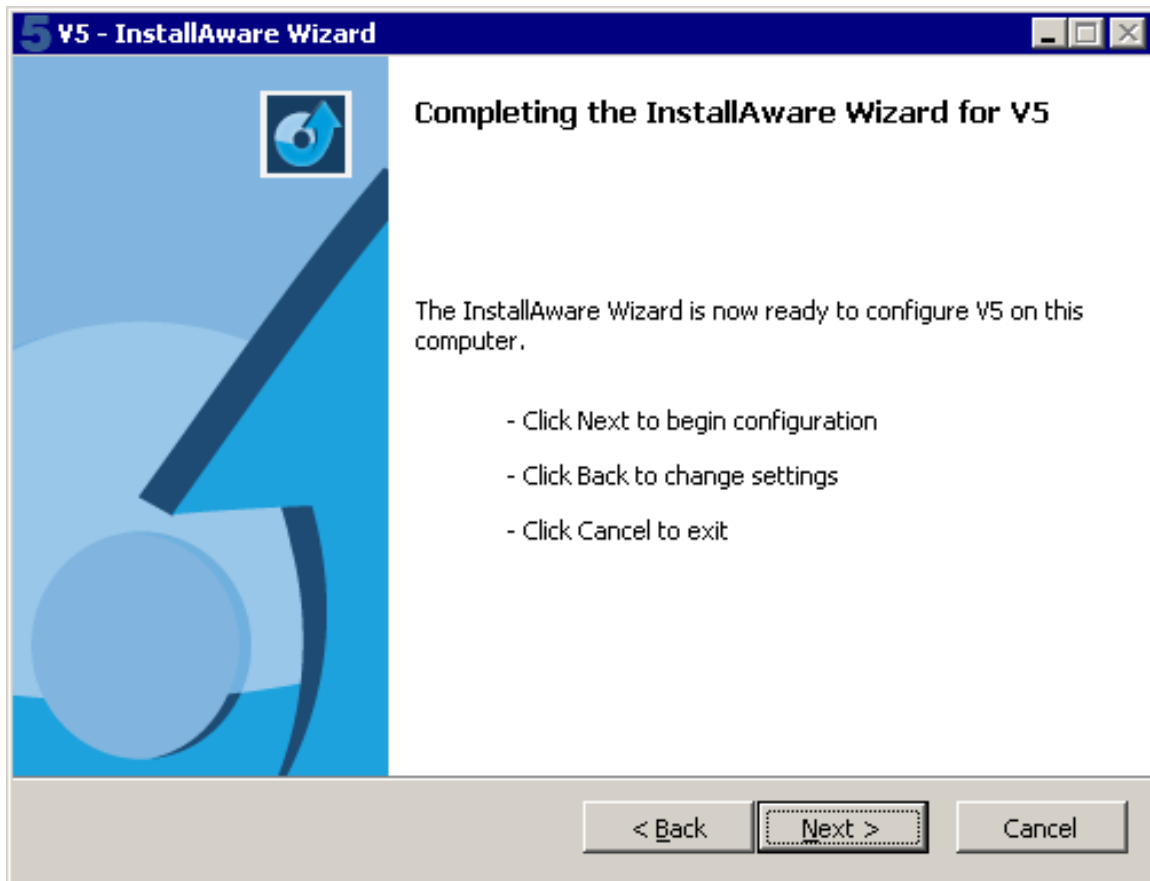
Click **Next >**.

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.



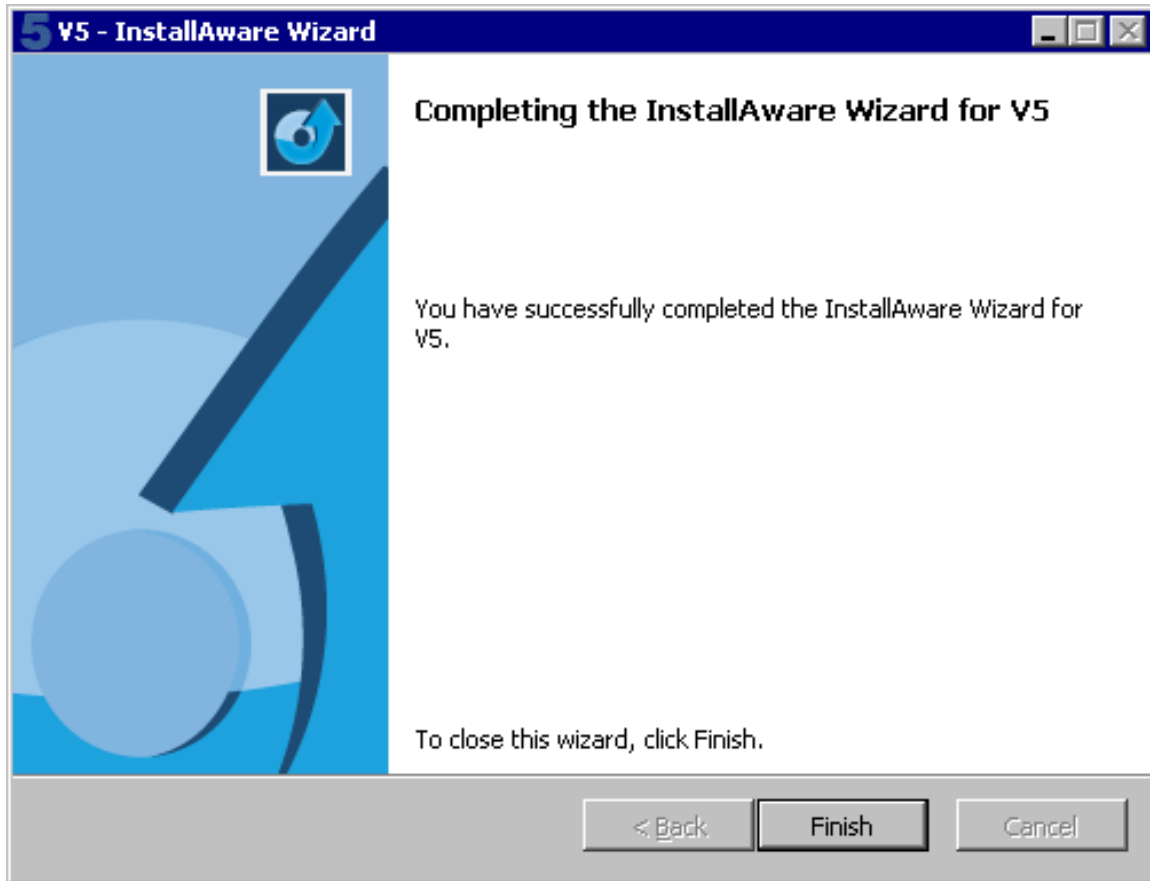
Click **Next >** to finalize the configuration and begin the installation.

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



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

8. After completing the installation, the wizard may give you the option to restart your computer by checking the **Restart now** box.



Note The option to restart your computer is typically only offered if FlashCut software has never been installed on this computer before.

9. Click **Finish** to close the wizard.

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 then there is a high likelihood that you will have to manually install the USB driver. When the hardware is connected to the computer with the USB cable and then powered on, the PC will try and find a driver for the device and chooses what it thinks makes the most sense. This means that the PC may use a driver from a wireless mouse by mistake, but if the software is installed first the PC will select the proper driver.

The USB driver will be automatically installed as long as the Flashcut CNC software has been successfully installed prior 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.

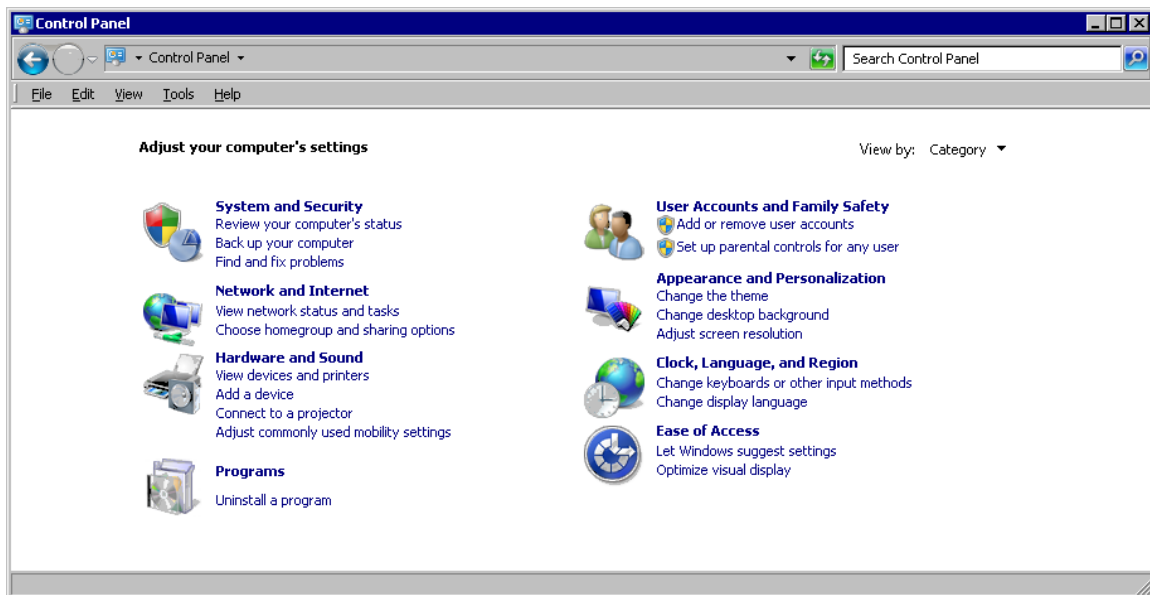
1. Make sure the FlashCut software is installed.
2. Connect the signal generator to the PC using an A-B USB cable.
3. Connect the power adapter to the signal generator.
4. Turn on the signal generator.

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

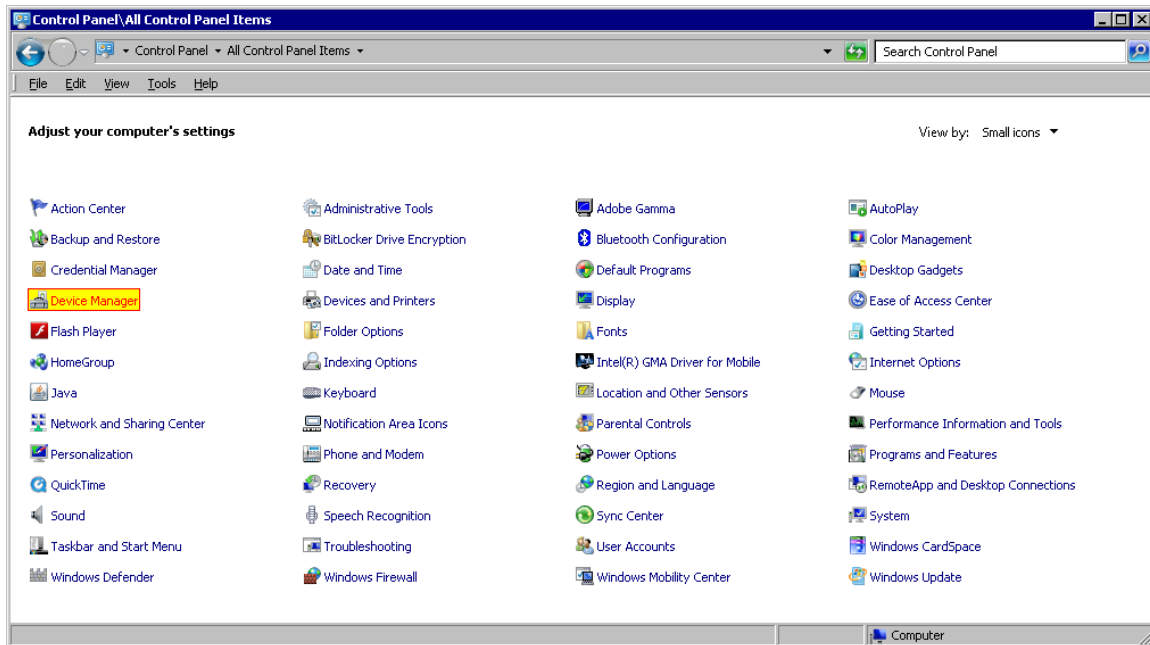
Windows 7

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.

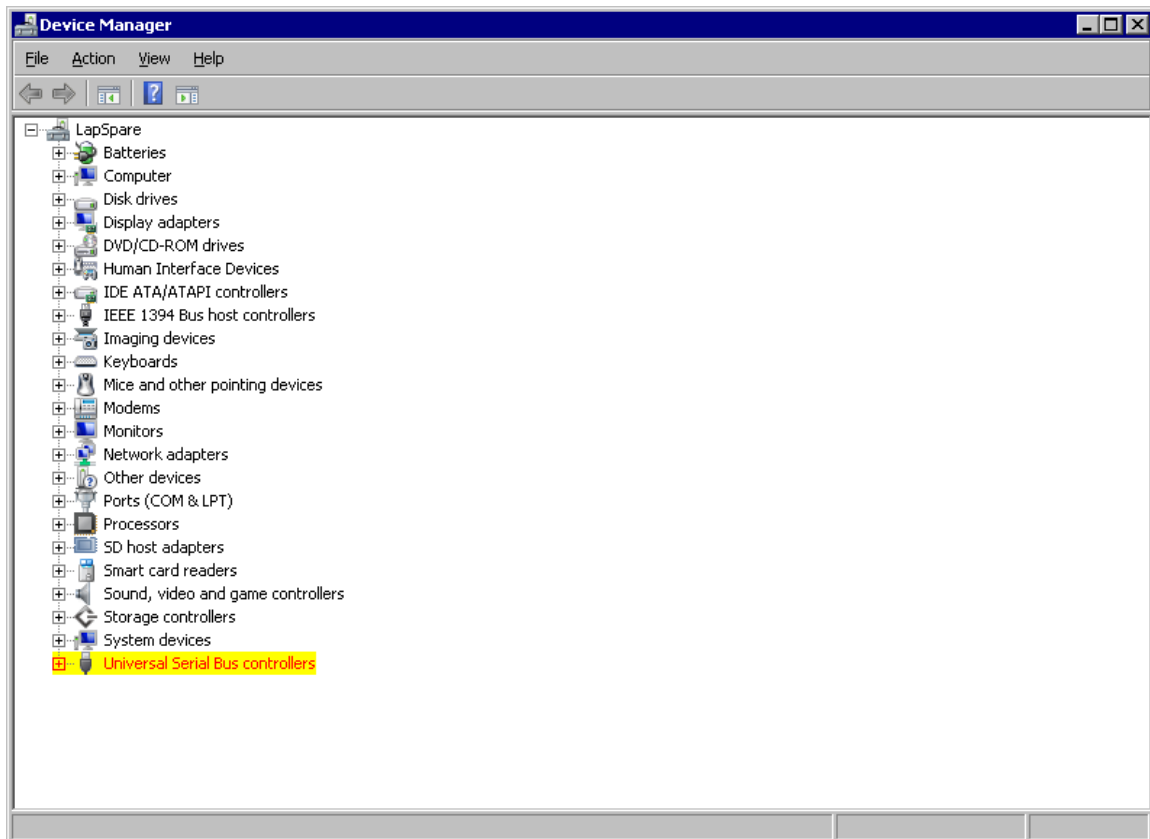
1. Click **Start ► Control Panel**. Windows displays the Control Panel screen.



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

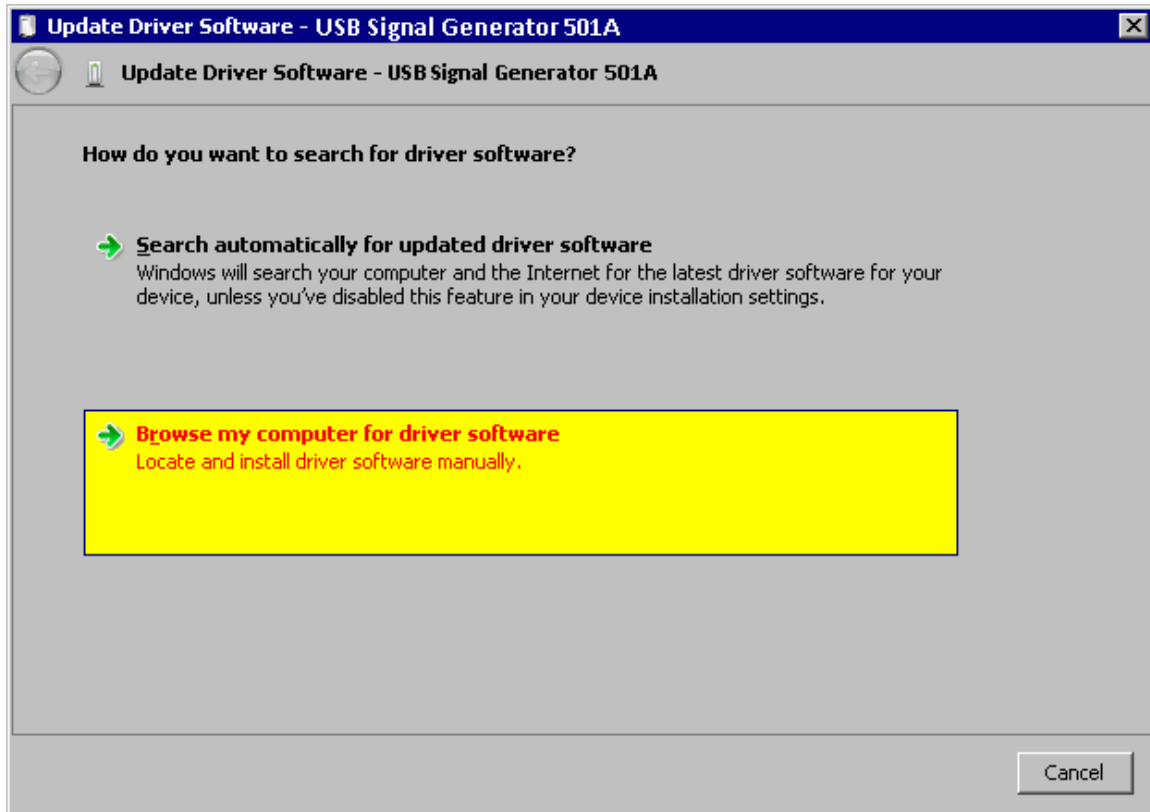


3. Click **Device Manager**. Windows displays the Device Manager screen.



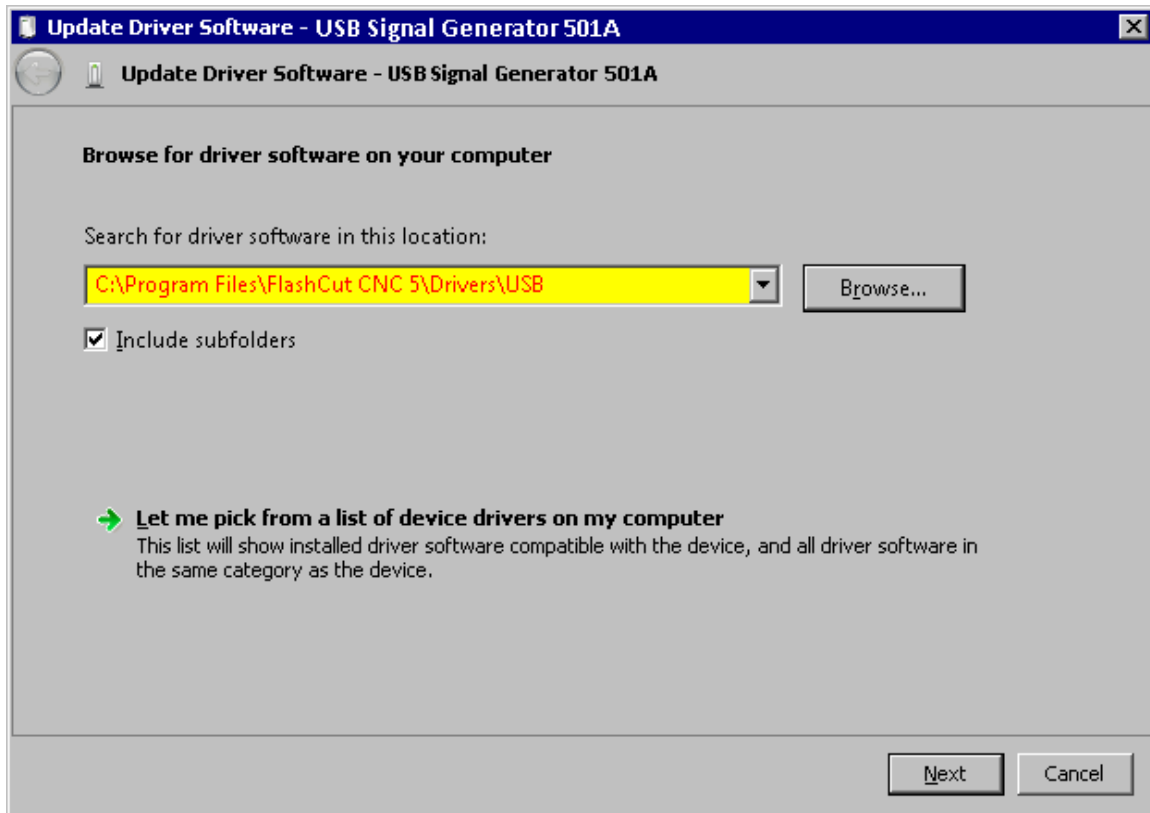
4. Expand the **Universal Serial Bus Controllers** listing.

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.

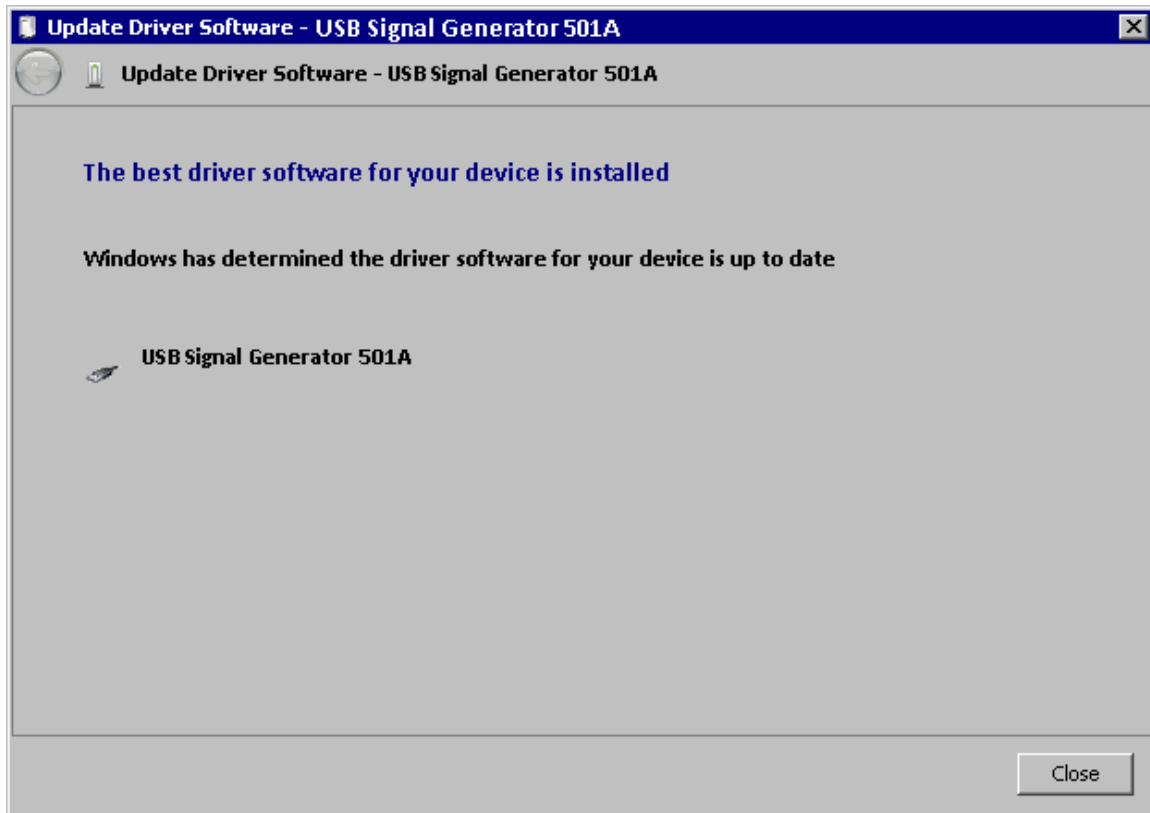


7. Click **Browse** and navigate to the following folder:

- **32-bit Windows 7**
C:\Program Files\FlashCut CNC 5\Drivers\USB
- **64-bit Windows 7**
C:\Program Files (x86)\FlashCut CNC 5\Drivers\USB

When you've 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**.

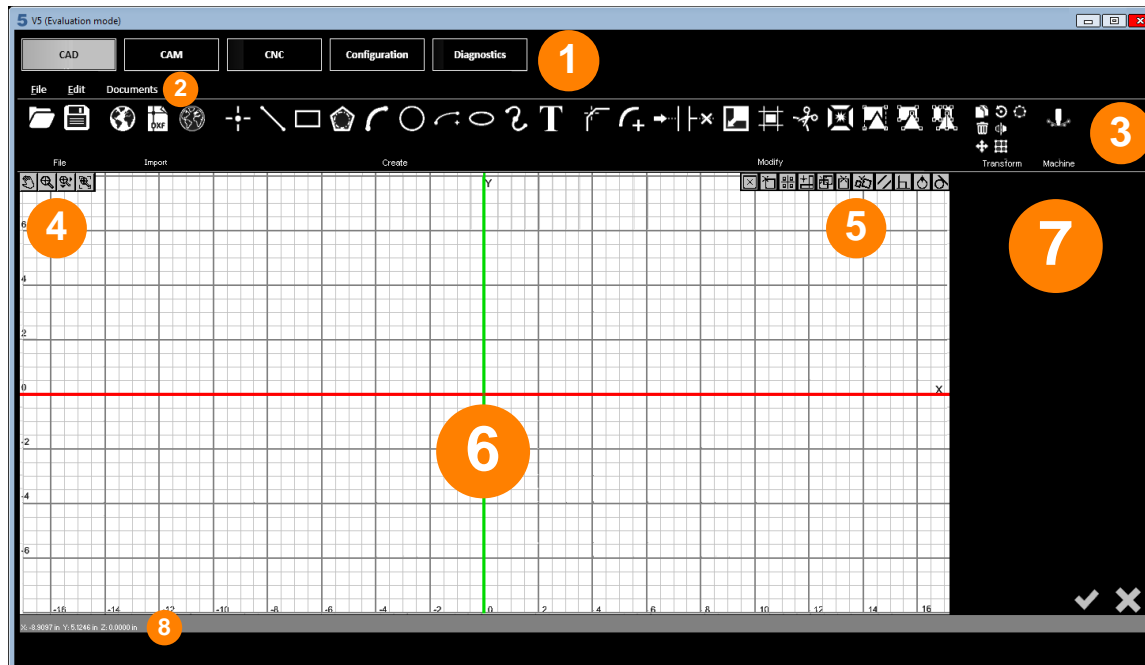
Windows 8

Windows automatically finds and installs the driver for the USB signal generator.

FlashCut CAD

FlashCut CAD features a two-dimensional CAD (Computer-Aided Design) editor for creating quick and detailed drawings. The FlashCut CAD should be used to draw the part only. The toolpath with kerf compensation, lead-ins, lead-outs and nesting are generated in FlashCut CAM.

The FlashCut CAD main screen is shown here. FlashCut CAD has many configuration options that can affect the appearance of the screen.



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

- | | |
|--------------------------------------|-------------------------------------|
| 1 1 <u>Tab</u> s | 5 5 <u>Snap</u> tools |
| 2 2 <u>Menu</u> bar | 6 6 <u>Drawing</u> workspace |
| 3 3 <u>Ribbon</u> | 7 7 <u>Parameters</u> area |
| 4 4 <u>Pan and zoom</u> tools | 8 8 <u>Status</u> bar |

Tabs

Use the tabs at the top of the screen to choose the type of tasks you'd like to do.



- CAD displays [FlashCut.CAD](#).
- CAM displays [FlashCut.CAM](#)
- CNC displays [FlashCut.CNC](#)
- Configuration displays the FlashCut [Configuration](#) interface
- Diagnostics displays the FlashCut [Diagnostics](#) interface

Menu bar

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



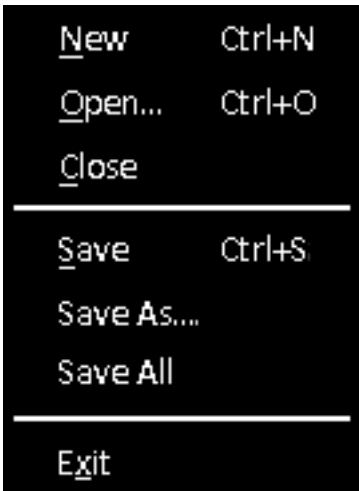
FlashCut CAD has the following menus:

- [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](#)
- [Exit](#)



New (Ctrl+N)

Creates a new CAD/CAM drawing in a new FlashCut CAD window. It does not close the current drawing.

Open (Ctrl+O)

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

Close

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

Save (Ctrl+S)

Saves the current CAD/CAM drawing using the existing file name and location. It will not save any other CAD/CAM drawing that is open. FlashCut will prompt you to create a file name and destination. FlashCut saves drawings to the CAD/CAM (*.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.

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.

Save all

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

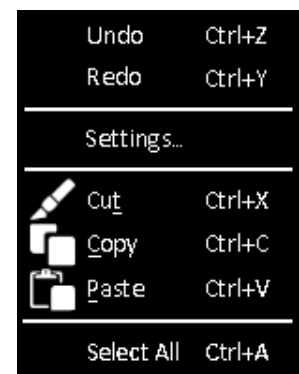
Exit

Closes FlashCut. 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.

Redo (Ctrl+Y)

Repeats the previous drawing action.

Settings...

Displays the drawing configuration controls. After changes are complete, click the green check mark to accept changes or the red X mark to reject changes.

Visible

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

Dynamic Grid

Toggle dynamic gridlines within the drawing window. These gridlines remain the same size despite zooming and panning motions.

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.

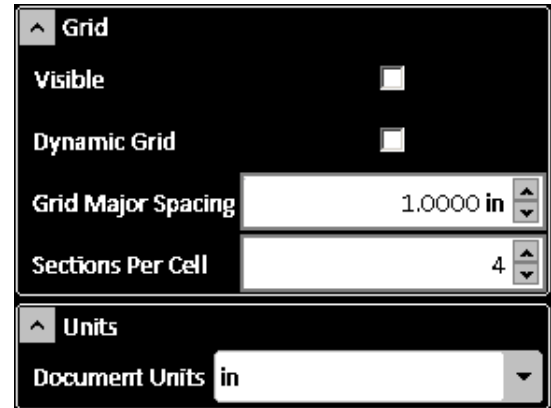
Sections Per Cell

Determines the number of minor horizontal and vertical gridlines in between major gridlines.

Document Units

Select the dimensional units of the CAD drawing. After changing, FlashCut gives you the option to either rescale or reinterpret any existing entities 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). You may choose the following units:

- mm – millimeters
- cm – centimeters
- m – meters
- in – inches
- ft – feet
- yd – yards



Cut (Ctrl+X)

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

Note Objects are selected by clicking on them with the selection arrow (which becomes available by pressing the **Esc** 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)

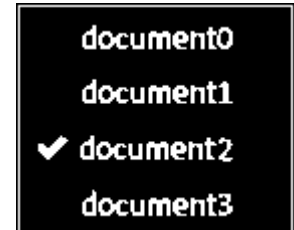
Pastes copied or cut features from the clipboard. The features will be centered at the cursor point and you will be able to maneuver them to a desired location. Clicking the mouse will paste the features permanently into the drawing.

Select All (Ctrl+A)

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

Documents menu

The **Documents** menu allows you to toggle between opened drawings. The checked document is open and available for editing. You may copy features from one drawing and paste them into another drawing.



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](#)
- [Modify tools](#)
- [Transform tools](#)
- [Machine tool \(send drawing to CNC\)](#)

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.

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 image – imports lines and arcs from a DXF CAD drawing



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

Import silhouette image

This tool takes an imported image and renders it as a series of closed line segments, forming a silhouette. Images with defined lines generally result in a cleaner drawing that requires fewer revisions. When selected, a number of options appear in the parameter window. After the drawing is the desired shape, click the green check mark to approve changes or the red X mark to discard the silhouette.

Choose image

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

Scale

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

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.

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 Image

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 Image**, FlashCut presents these options:

- **Background color** – Pick which corner of the image that is to be used to sample the background color. The color sampled from the specified corner will be used to trace the boundary of the desired silhouette.
- **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.

- **Color Tolerance** – This tolerance is used to determine the boundary between the background and the silhouette. How close does a color have to be to the background color to be included in the background and later removed.

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.

Segmentation quality

FlashCut CAD automatically breaks curves in the drawing into small, separate line segments. Select how finely the program will divide curves. When increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing file, the program's memory usage, and the time to manufacture the final part.

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 tighter tolerance replicates the image very closely.

Despeckling

Reduces the number of small dots that appear in the image. The larger the number, the lower the dot tolerance, causing fewer dots to appear in the imported silhouette.

Smoothing Factor

Reduces the magnitude of small jagged features on an entity. The larger the factor the smoother the result.

Corner threshold

Select the tendency of FlashCut CAD to generate rounded corners at line vertices. Increasing this parameter will increase the number of rounded corners in the drawing.

Inverted

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

Import DXF image

This tool imports a two dimensional DXF file and renders it in the drawing. When selected, a number of options appear in the parameter window. After the drawing is the desired shape, click the green check mark to approve changes or the red X mark to discard them.

Choose image

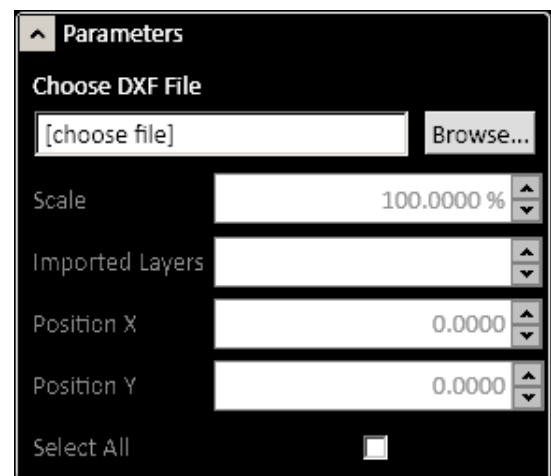
Click **Browse...** to select a DXF file from the computer. Click **Open** to call up the desired image.

Scale

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

Imported layers

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



Position X

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

Position Y

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

Select all

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

Import centerline image

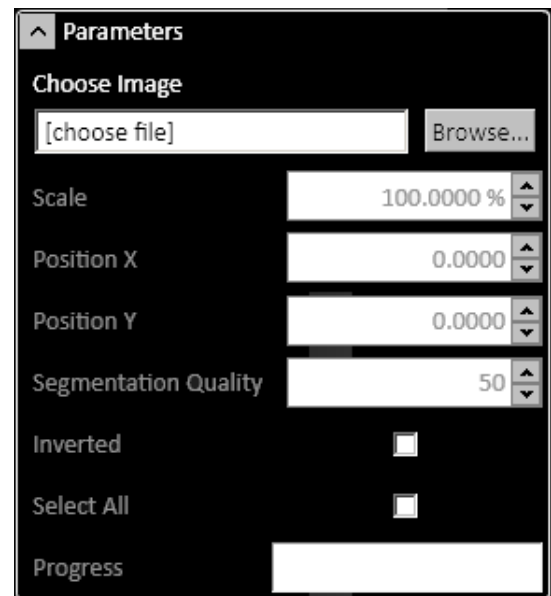
This tool imports a bitmap image and renders each feature as a single toolpath line down the center of the feature. Images with defined lines generally result in a cleaner drawing that requires fewer revisions. When selected, a number of options appear in the parameter window. After the drawing is the desired shape, click the green check mark to approve changes or the red X mark to discard the imported design.

Choose image

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

Scale

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



The screenshot shows a 'Parameters' dialog box with a dark background. At the top, there's a 'Choose Image' section with a text input field containing '[choose file]' and a 'Browse...' button. Below this are several settings: 'Scale' is set to '100.0000 %' with up/down arrows; 'Position X' is set to '0.0000' with up/down arrows; 'Position Y' is set to '0.0000' with up/down arrows; 'Segmentation Quality' is set to '50' with up/down arrows; 'Inverted' has an unchecked checkbox; 'Select All' has an unchecked checkbox; and 'Progress' has a progress bar.

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

FlashCut CNC automatically breaks curves in the drawing into small, separate line segments. Select how finely the program will divide curves. When increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing file, the program's memory usage, and the time to manufacture the final part.

Inverted

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

Select all

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

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. Choosing the **For Construction** option changes the solid lines of the element into a dotted line. Construction lines are meant to aid in drawing and are not utilized by FlashCut CAM and FlashCut CNC. After changes are complete, click the green check mark to approve changes or the red X mark to discard changes. These are the tools that are available on the **Create** portion of the ribbon:

Note 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.

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

Point



Click on the drawing to create a new point at that location. You may edit coordinates in the parameter window.

Line



Click on the drawing to place the first endpoint of a line. Move the cursor to the desired location of the second endpoint and click the mouse to place the line. You may edit start and endpoint coordinates, length, and angle in the parameter window. Choosing **Set Length** or **Set Angle** locks in the parameters until you exit from the tool.

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. Selecting which method will be used, and then click the rectangle points in the drawing window. You can manually enter point coordinates as well.

Polygon



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. FlashCut CNC enables you to select the number of vertices using the Count setting as well as the option to choose whether the shape will be inscribed or

circumscribed within a construction circle. Choosing the Construction Circle option on will cause this circle to remain in the drawing after the polygon is created.

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. Choosing Forward on or off reverses the direction of the arc. The 3 point arc enables you to select two points on the curve, followed by a third point that determines the degree of curvature in between these points.

Circle



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

Elliptic Arc



Click in the drawing window to select a centerpoint 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. Finally, select a start point and then an end point to the arc.

Ellipse



Click in the drawing window to select a centerpoint 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.

Spline



Click the start point of the curve and then select subsequent points through which the spline will pass. When the spline is of desired shape, press the green check mark in the parameter window or press **Enter** to insert it into the drawing or, press the Esc key to reject the spline and exit the tool.

Text



FlashCut CNC enables you to enter text into drawings. In the parameter window, type the desired message in the Text field, and then select the font, alignment, size, Kerning offset (letter spacing), and angle of the text in the parameter window.

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.

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 approve changes or the red X mark to discard changes. These are the tools that are available on the **Modify** portion of the ribbon:

- [Apply Chamfer](#)
- [Trim Tool](#)
- [Reversed](#)
- [Apply Fillet](#)
- [Scale Entities](#)
- [Cut Tool](#)
- [Extend Tool](#)
- [Apply Offset](#)
- [Explode Entities](#)

Apply Chamfer



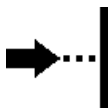
Select two intersecting lines or their intersection point, and then click this tool to add a sloped chamfer to this edge. You can scale the size of the chamfer by dragging the mouse towards or away from the chamfer, or in the parameter window, type the distances from the intersection in both directions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



The extend tool enables you to continue a line or a curve until it intersects with another object in the drawing. 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. Click the mouse to finalize the extension.

Trim Tool



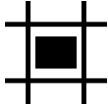
The trim tool enables you to trim a line or a curve at its intersection with another object in the drawing. Click the trim tool, and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear. Click the mouse to finalize the trim.

Scale Entities



Select a feature or group of features to be scaled. 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



The offset tool duplicates selected features and positions them at the specified distance away from the original feature. To offset a feature, select that feature and then choose the Apply Offset button. You may then select parameters for the offset in the parameter window.

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.



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.

Use Lines to Close Wires

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

Cut Tool

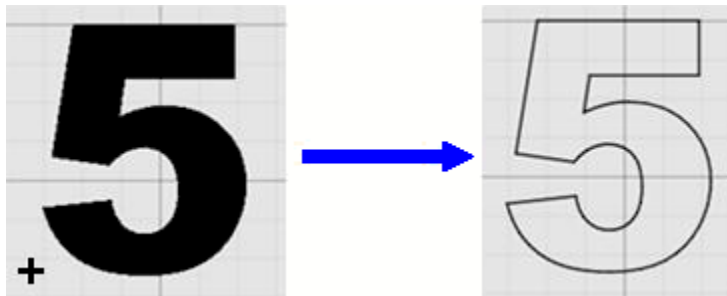


The cut tool divides a segment into separate segment. Click the mouse on the part of the segment to be split. The segment will separate at that point. You can verify this by hovering the cursor over the feature. Only a part of it will highlight if the cut worked properly.

Explode Entities



Transforms a feature that is solid and not composed of lines (such as a font) into individual line segments. This is necessary for toolpath generation of any feature that is solid. Select a solid feature then choose **Explode Entities**. See the example below.



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. First, select the features to be modified, then select the transform tool that will be used. You may edit parameters for the transformation in the parameter window. After transformations are complete, click the green check mark to approve changes or the red X mark to discard changes. These are the tools that are available on the **Transform** portion of the ribbon:



- [Copy entities](#)
- [Delete selected objects](#)
- [Move entities](#)
- [Rotate 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 a reference point about which the features will be moved, and then select the new location for the features.

Rotate entities



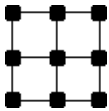
FlashCut CNC offers two modes for rotating features. The 3 Points rotation prompts you to select a center point, a point near the feature, and a final point to which the feature will rotate. The Point and Angle rotation requires you to select a central rotation point, and then enter a rotation amount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

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.

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

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

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 approve changes or the red X mark to discard changes.

Circular pattern tool



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

- The **Center X** and **Center Y** options specify the point about which the feature will be rotated. Alternately, click and drag the blue center point of the rotation.

Parameters	
Center X	0.0000 in
Center Y	0.0000 in
Spacing Mode	<input type="checkbox"/>
Angle	360.0000 °
Count	1

- 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.

Machine tool (send drawing to CNC)



Click the **Machine** icon (**Cut the Objects**) to create a G-code program directly from the CAD drawing using the default kerf compensation, lead in, lead out, and cutting parameters from FlashCut CAM. The generated G-code file gets directly loaded into FlashCut CAM. This is a quick and convenient way to cut a part, if all of the CAM parameters are unchanged from the last time a toolpath was generated. Use FlashCut CAM to change the CAM parameters.

Pan and zoom tools



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



Pan

The pan tool allows you to click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window also allows you to pan without the use of the pan tool.



Zoom Extent

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



Zoom

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



Zoom to Fit

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

Snap tools



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



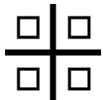
Features connect to the center of elements like circles.

Endpoint snap



Features connect to the endpoints of elements like lines.

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 midpoint of elements like lines.

Nearest snap



Features connect with the element closest to the cursor.

Parallel snap



Features align parallel with elements of the drawing like lines. A blue dashed line indicates that the feature is parallel to the red highlighted element in the drawing.

Perpendicular snap



Features align perpendicular with elements of the drawing like lines. A blue dashed line indicates that the feature is perpendicular to the red highlighted element in the drawing.

Quadrant snap



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

Tangent snap



Features connect tangentially to elements in the drawing like circles.

Drawing workspace

The drawing workspace is the area where you create and modify your CAD drawing.

Parameters area

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

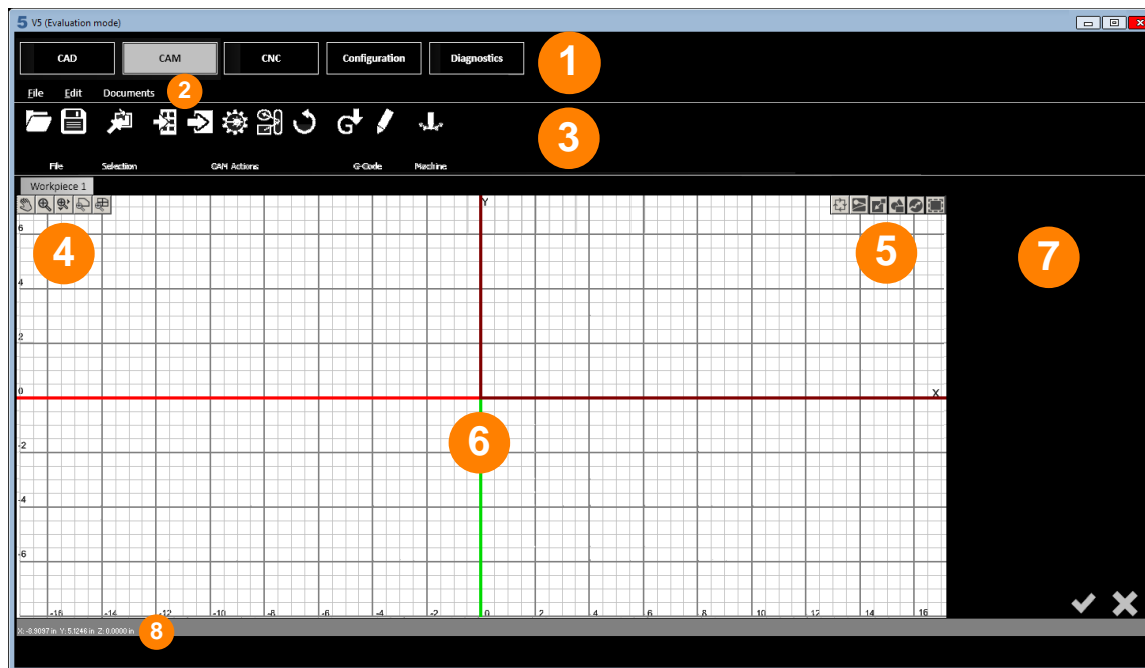
Status bar

Reports the real time position of the cursor, in addition to providing instructions to the user for the expected input of the currently selected CAD tool.

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. FlashCut CAM has many configuration options that can affect the appearance of the screen.



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

- | | |
|------------------------------------|---------------------------------------|
| 1 <u>Tab</u> | 5 <u>Display options tools</u> |
| 2 <u>Menu bar</u> | 6 <u>Drawing workspace</u> |
| 3 <u>Ribbon</u> | 7 <u>Parameters area</u> |
| 4 <u>Pan and zoom tools</u> | 8 <u>Status bar</u> |

Tabs

Use the tabs at the top of the screen to choose the type of tasks you'd like to do.



- CAD displays [FlashCut CAD](#)
- CAM displays [FlashCut CAM](#)
- CNC displays [FlashCut CNC](#)
- Configuration displays the FlashCut [Configuration](#) interface
- Diagnostics displays the FlashCut [Diagnostics](#) interface

Menu bar

This area contains menu headings for system commands. Hotkey commands are listed next to their respective functions. Accelerator keys are displayed when you press the **{Alt}** key.



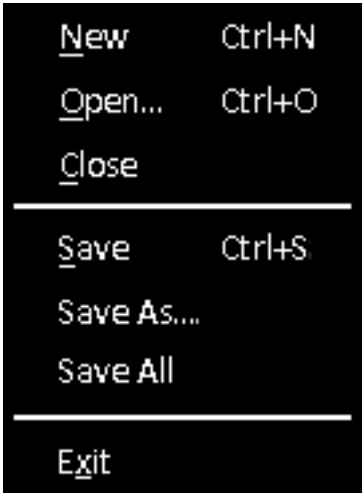
The menu bar has these menus:

- [File menu](#)
- [Edit menu](#)
- [Documents menu](#)

File menu

The **F**ile menu presents the following commands:

- [New \(Ctrl+N\)](#)
- [Open \(Ctrl+O\)](#)
- [Close](#)
- [Save \(Ctrl+S\)](#)
- [Save as](#)
- [Save all](#)
- [Exit](#)



New (Ctrl+N)

Creates a new CAD/CAM drawing in a new FlashCut CAD window. It does not close the current drawing.

Open (Ctrl+O)

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

Close

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

Save (Ctrl+O)

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.

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.

Save all

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

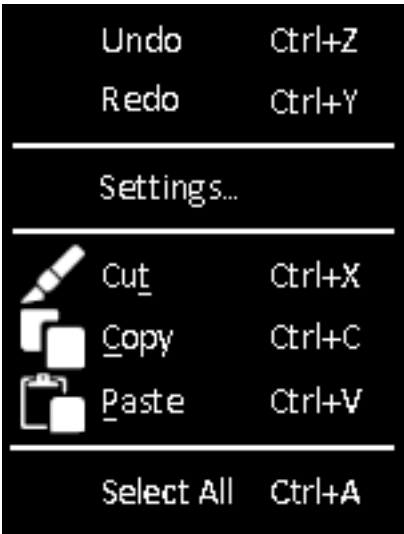
Exit

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

Edit menu

The **Edit** menu presents 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.

Redo (Ctrl+Y)

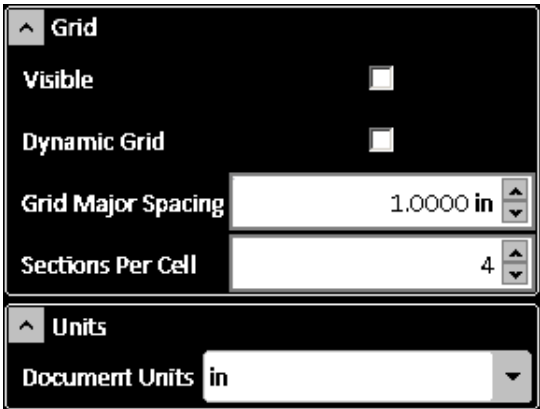
Repeats the previous drawing action.

Settings...

Displays the drawing configuration controls. After changes are complete, click the green check mark to accept changes or the red X mark to reject changes.

Visible

The **Visible** checkbox enables/disables the visibility of the grid in the drawing workspace.



Dynamic Grid

The **Dynamic Grid** checkbox enables and disables dynamic gridlines within the drawing workspace. These gridlines remain the same size despite zooming and panning motions. To keep the gridlines the same size despite zooming and panning motions, disable dynamic grid.

Grid Major Spacing

Changes the size of the major gridlines when **Dynamic Grid** is not toggled. 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.

Document Units

Select the dimensional units of the CAD drawing. After changing, FlashCut gives you the option to either rescale or reinterpret any existing entities 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). You may choose the following units:

- mm – millimeters
- cm – centimeters
- m – meters
- in – inches
- ft – feet
- yd – yards

Cut (Ctrl+X)

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

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.

Note Objects are selected by clicking on them with the selection arrow (which becomes available by pressing the **Esc** 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.

Select All (Ctrl+A)

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

Documents menu

The **Documents** menu allows you to toggle between opened drawings. The checked document is open and available for editing. You may copy features from one drawing and paste them into another drawing.



Ribbon

The FlashCut CAM ribbon has five types of command icons:

- **File tools**
- **Selection**
- **CAM Actions**
- **G-code**
- **Machine (send drawing to CNC)**



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)



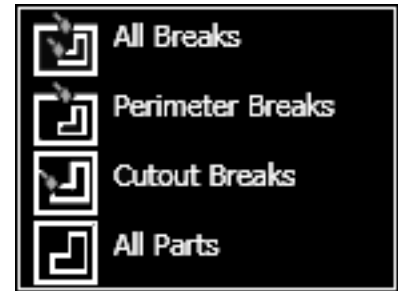
Saves the current CAD/CAM drawing using the existing file name and location. It will not save any other CAD/CAM drawing that is open. FlashCut will prompt you to create a file name and destination. FlashCut saves drawings to the CAD/CAM (*.cadcaml) format. These files represent CAD drawings readable by FlashCut CAD version 5 or later. This command is unavailable when no recent changes have been made.

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.



All Breaks

Selects all available breaks in the document, this includes perimeter breaks and cutout breaks

Perimeter Breaks

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

Cutout Breaks

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

All Parts

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

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.

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:



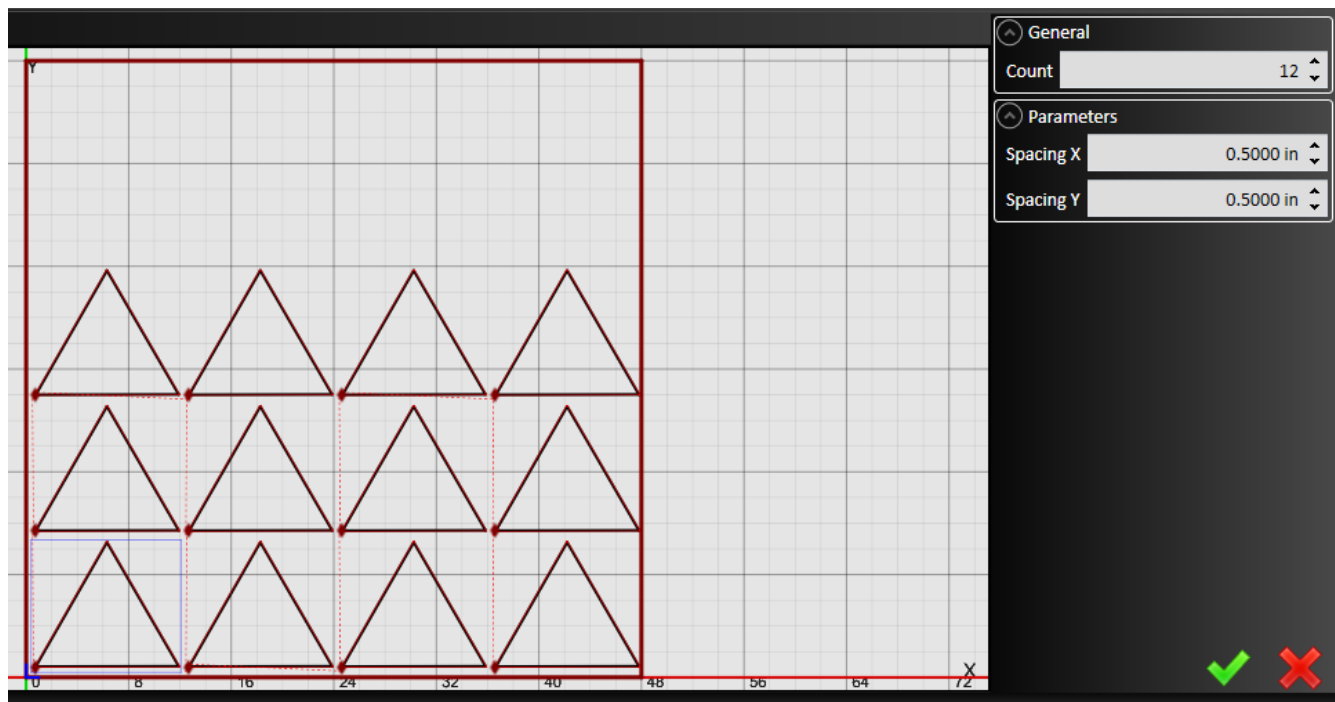
- [Grid Nesting](#)
- [TrueShape Nesting](#)
- [Plasma Settings](#)
- [Sequence tool](#)

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 12 triangles are grid nested.

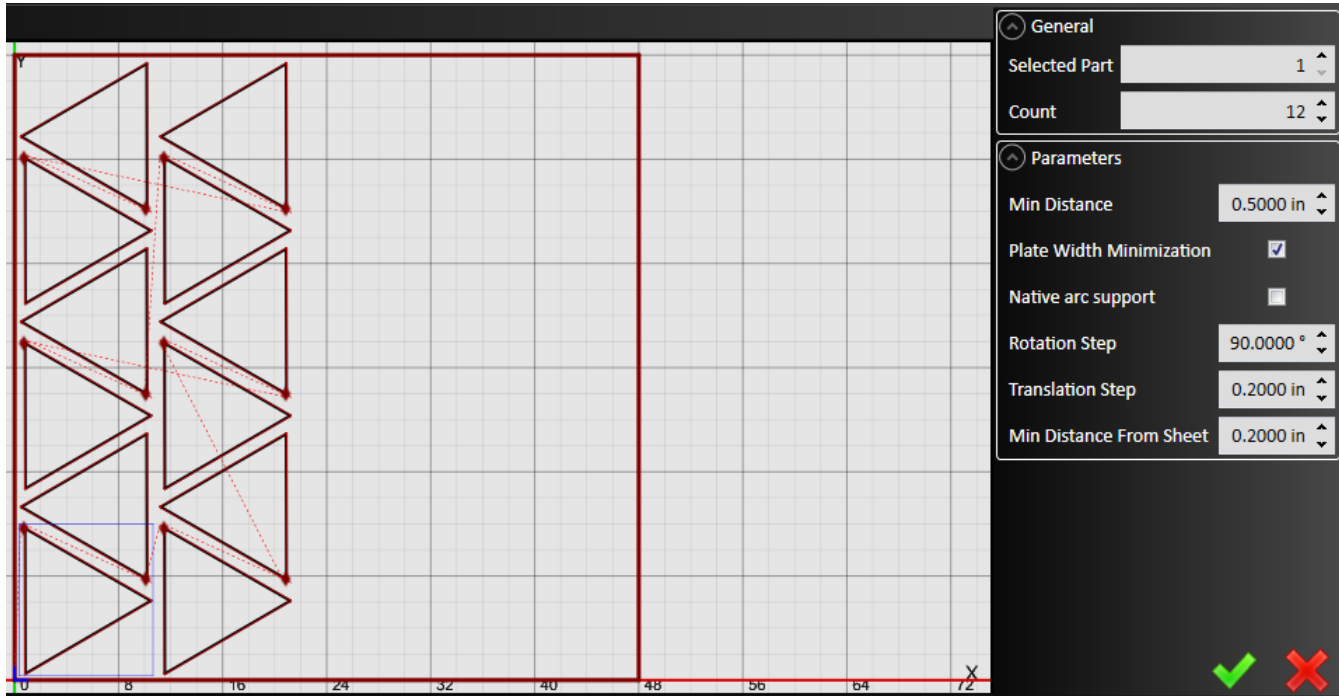


TrueShape Nesting



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. 12 triangles are nested below using TrueShape Nesting. Notice the material usage is much lower for TrueShape nesting as it is for

the same number of triangles using Grid nesting. The parameters for TrueShape Nesting are described below.



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.

Min Distance

Determines the minimum distance between compensated toolpath, lead-ins and lead-outs.

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.

Translation Step

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.

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.

Plasma Settings



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

Select the model of the plasma torch being used then select the nozzle, material type, thickness and desired quality setting. FlashCut automatically loads in the manufacturer's recommended plasma settings. The **Safe Z Level** number indicates the lowest height at which the torch will make rapid moves between cutting features or parts. You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, click the green check mark to approve or the red X mark to discard.

Torch

Select your plasma torch type. The parameters will automatically adjust to match your selection. Note that PowerMax® is a registered trademark of the Hypertherm® Corporation.

Nozzle

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

Material Type

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

Thickness

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

Parameters	
Torch	Powermax125
Nozzle	125A Shielded
Material Type	Mild Steel
Thickness	1/4 in
Kerf Width	0.0890 in
Quality Level	Best Quality
Feedrate	188.0000
Safe Z Level	0.2500 in
Corners	Sharp
Cut Height	0.1800 in
Pierce Height	0.3600 in
Pierce Delay	0.2000 in
Set Point	158.0000 in
Amperage	NA
Workpiece Width	48.0000 in
Workpiece Length	48.0000 in

Note Each of the following parameters will be automatically adjusted when the four conditions above are specified. However, they can also be input manually:

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.

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.

Safe Z Level

Input the height to which the torch will return after each cut. It is a safe plane on which the torch can move.

Corners

Toggle whether the corners on the toolpath will be sharp or rounded. This may take a short time to render the toolpath.

Cut Height

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

Pierce Height

Specify the height at which the torch will initially pierce before cutting along the toolpath.

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.

Set Point

This is the target voltage for the THC system. The set point value can be found in the operators manual of the plasma torch and is dependent on the material type, thickness and torch settings to name a few. When sampling is enabled, the system will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage

Displays the amperage at which the torch will cut.

Note The only way to physically change the amperage is on the plasma unit itself.

Workpiece Width

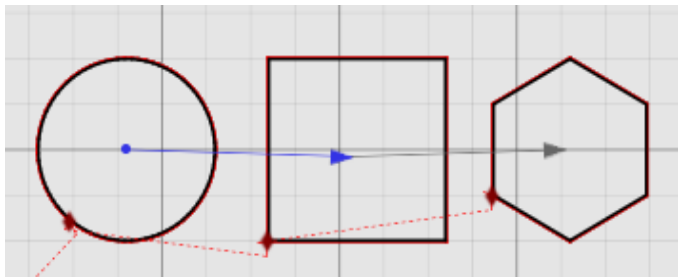
Specify the width of the sheet being cut. This will change the size of the material in the drawing window as well as any nesting settings.

Workpiece Length

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

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.



Machine (send drawing to CNC)



When CAM specifications are complete and accurate, click the **Machine** button to send them to FlashCut CNC. This automatically writes a G-code program for the machine to follow, which you may modify in FlashCut CNC prior to machining the part.

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 a tool to change the cursor's function.



Pan

The pan tool allows you to click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window also allows you to pan without the use of the pan tool.



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 to zoom to fill the workspace with the workpiece.

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.

The following display options are available:

- [Cutting directions](#)
- [Kerf](#)
- [Material](#)
- [Parts](#)
- [Rapid moves](#)
- [Tool paths](#)

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

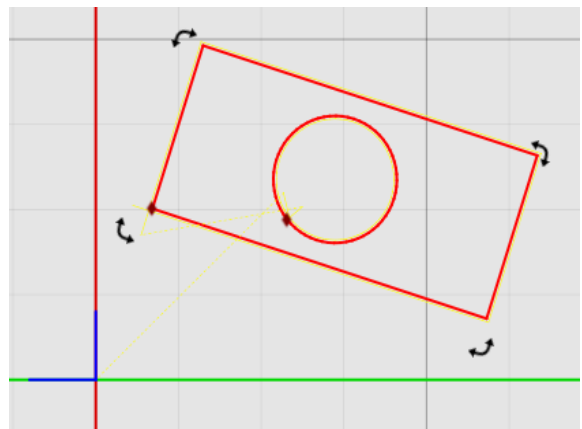
This section discusses the following topics:

- [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. When parameters are correct, press the green check mark to approve changes or the red X mark to discard them. 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.

The lead-in, lead-out settings are described 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.

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 below are displayed.

Line Length

Select the length of the leading line.

Line Angle

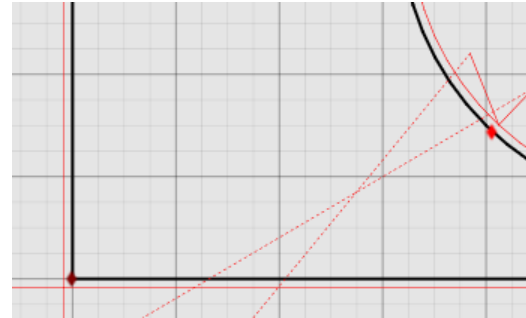
Select the angle that the leading line makes with the toolpath. The angle is measured clockwise.

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.



General

Overburn
0.0000

LeadIn

Type

☐ No Lead

☒ Line

☐ Arc

Line Length
0.3750 in

Line Angle
0.0000 °

LeadOut

Type

☐ No Lead

☒ Line

☐ Arc

Line Length
0.2500 in

Line Angle
0.0000 °

☒
☒

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](#).



Parameters area

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

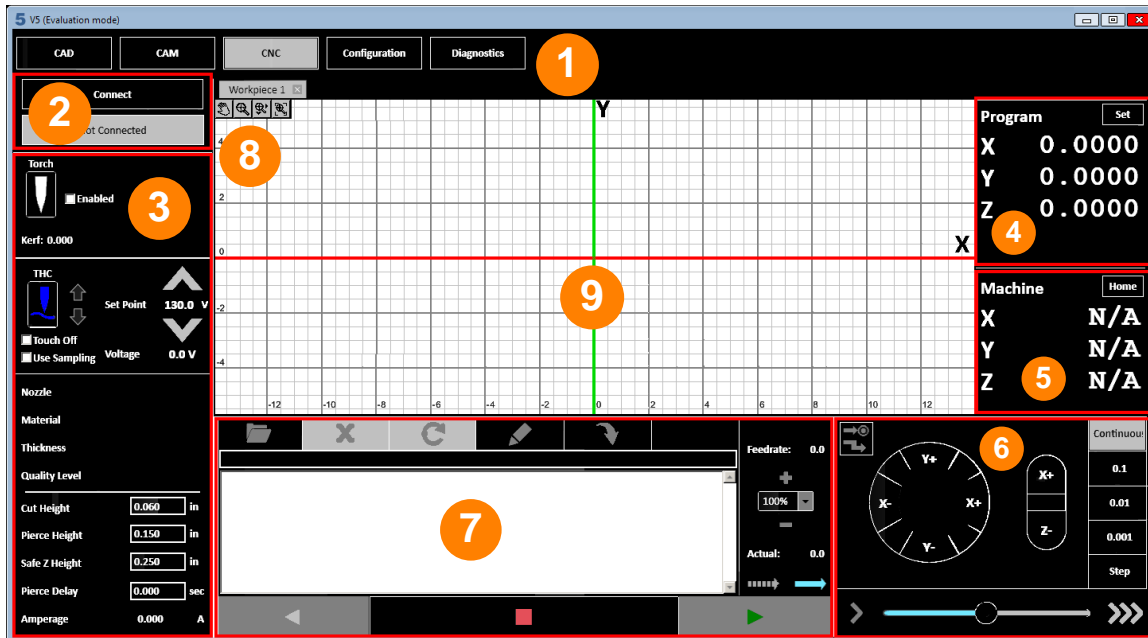
Status bar

Reports the real time position of the cursor, in addition to providing instructions to the user for the expected input of the currently selected CAD tool.

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 *FlashCut CNC Programming Reference* manual.

The main screen is shown below. An explanation of each area of the screen follows. Note that FlashCut has many configuration options that can affect the appearance of the screen.



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

- | | |
|---|--------------------------------------|
| 1 Tabs | 6 Jog control panel |
| 2 Signal generator status | 7 G-code window |
| 3 Plasma torch settings | 8 Pan and zoom tools |
| 4 Program coordinates panel | 9 Drawing workspace |
| 5 Machine coordinates panel | |

Tabs

Use the tabs at the top of the screen to choose the type of tasks you'd like to do.



- CAD displays [FlashCut.CAD](#)
- CAM displays [FlashCut.CAM](#)
- CNC displays [FlashCut.CNC](#)
- Configuration displays the FlashCut [Configuration](#) interface
- Diagnostics displays the FlashCut [Diagnostics](#) interface

Signal generator status

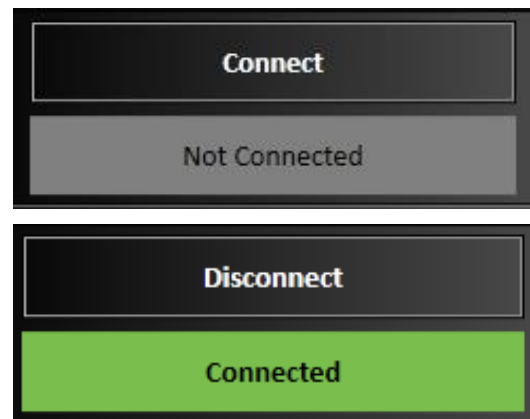
Ensure that the signal generator (CNC controller) is securely connected via USB cable to your PC, and you've completed the USB driver installation (see [Installing the USB driver](#) for instructions).

Turn on the signal generator.

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

When properly connected, the **Connect** button changes to a **Connected** status indicator. Press **Disconnect** prior to unplugging the signal generator.


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



Plasma torch settings


After connecting to the signal generator, the machine preferences will be displayed in this panel. Those in the bottom two boxes are indicators, while those in the top box may be edited.

- **Torch** – Clicking the torch icon will open up a dialog box prompting you to turn on the output line. Selecting **Yes** will proceed to turn on the output line (torch).
- **THC (Torch Height Control)** – Select the THC icon to toggle torch height control on or off.
- **Set Point** – Target voltage for the THC algorithm to track. The set point voltage is either read from a table from parameters in the Plasma Torch Settings panel, or from the sampled voltage. It can be adjusted on the fly by hitting the up and down arrows.
- **Touch Off** – The torch will touch the material in order to establish a zero point on the machine coordinates. Toggling **Touch Off** on will cause the torch to touch down to reestablish the zero point between rapid and feedrate moves.
- **Use Sampling** – After moving to the precise cut height, the THC system will begin sampling the voltage. It will use this sampled voltage as the Set Point voltage
- **Voltage** – Displays the actual arc voltage between the plasma nozzle and the workpiece.
- **Nozzle** – This is a torch consumable, providing this information allows the software to properly generate the toolpath.
- **Material** – Enter the stock material, this will provide necessary information for the software to properly offset the toolpath.
- **Thickness** – The thickness is required to properly offset the kerf and set the feedrate of the program. The thicker the material the slower the torch must go to cleanly cut through the material.
- **Quality Level** – Torch manufactures often provide different cut parameter profiles, specify the best quality or the fastest cut.
- **Cut Height** – The cut height is 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, if applicable, with a sampling height.
- **Pierce Height** – The pierce height is 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, if any, expires.
- **Pierce Delay** – This is a delay intended to allow the torch time to pierce completely through the stock material. The delay begins when the torch is commanded to fire and there is no motion in any direction until the delay expires.



Torch

Kerf: 0.055



THC

Set Point **131.7 V**

☒ Touch Off

☒ Use Sampling

Voltage **8.0 V**

Nozzle **45A Shielded**

Material **Mild Steel**

Thickness **14GA**

Quality Level **Best Quality**

Cut Height **0.060**

Pierce Height **0.150**

Pierce Delay **0.200**

Safe Z Height **0.250**

Amperage **0.000**

- **Safe Z Height** – Also commonly referred to as the clearance plane, the safe Z height should be defined such that the torch will retract above any parts, stock, fixtures, or other obstructions that may exist in the machine envelope. The safe height is not included in the G-Code program directly but is called automatically before any rapid, or G00, command.
- **Amperage** – The operating amperage of the torch.

Program coordinates panel

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.

Beneath the coordinates are two menus of instruction that affect the program coordinates:

- **Move** – Moves the tool a specified location. Program zero marks the point where all three axis coordinates equal zero. **To Program Zero** moves the X and Y position to zero and then moves the tool vertically upward in the Z direction to a height that you may specify in the configuration settings.
- **Set** – Sets program zero points along each axis, as selected from the dropdown menu. **Zero All** will zero each program axis. The **Sense Z Zero** command 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.

Program	
X	0.0000
Y	0.0000
Z	0.0000
<div>Move</div> <div>Set</div>	

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.

Beneath the coordinates are two menus of instruction that affect the program coordinates:

- **Move** – Moves the tool to a specified location.
 - **To Home** moves the tool to the determined home position.
 - **To Torch Maint. Position** and **To Material Load** position move the tool to these respective positions, which you may enter in the configuration settings.
 - **To Point...** enables you to specify a point in machine coordinates to which the tool is moved.

Machine	
X	0.0000
Y	0.0000
Z	0.0000
<div>Move</div> <div>Home</div>	

- **Home** – After connecting the signal generator to FlashCut, it is necessary to seek out and set the home point, or Machine Zero, of the tool. You may home axes individually or all at once by using the **Home All** command and pressure switches at the ends of each axis. To save time, 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.

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.

It is necessary to home the axes normally first. The **Zero** command sets the current point as the origin of the machine coordinates. **Clear** closes the machine coordinate system. This displays **N/A** on each axis and requires you to reestablish a machine zero point.

Jog control panel

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

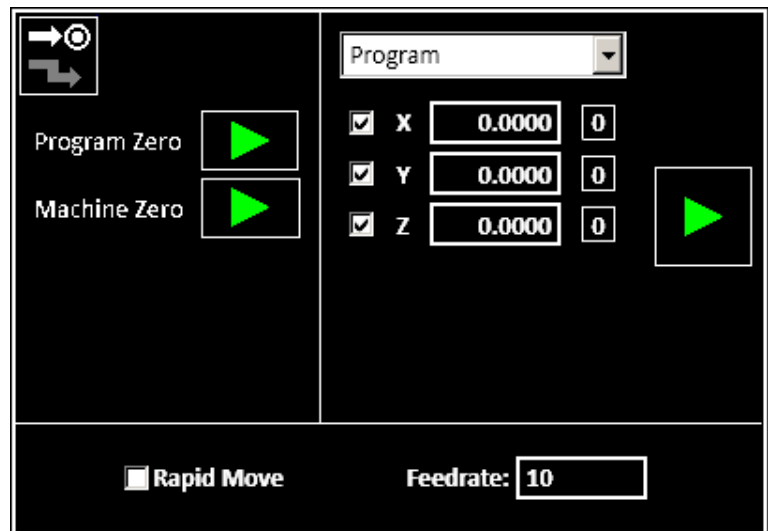
- **Jog-to-point**
- **Manual jogging**

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

Jog-to-point

Jog-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.

Manual jogging

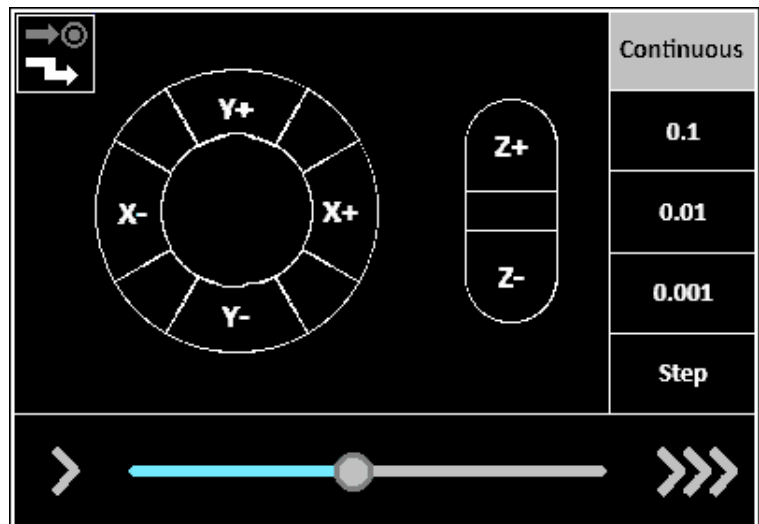
The jog control panel provides these controls for manually positioning all axes.

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

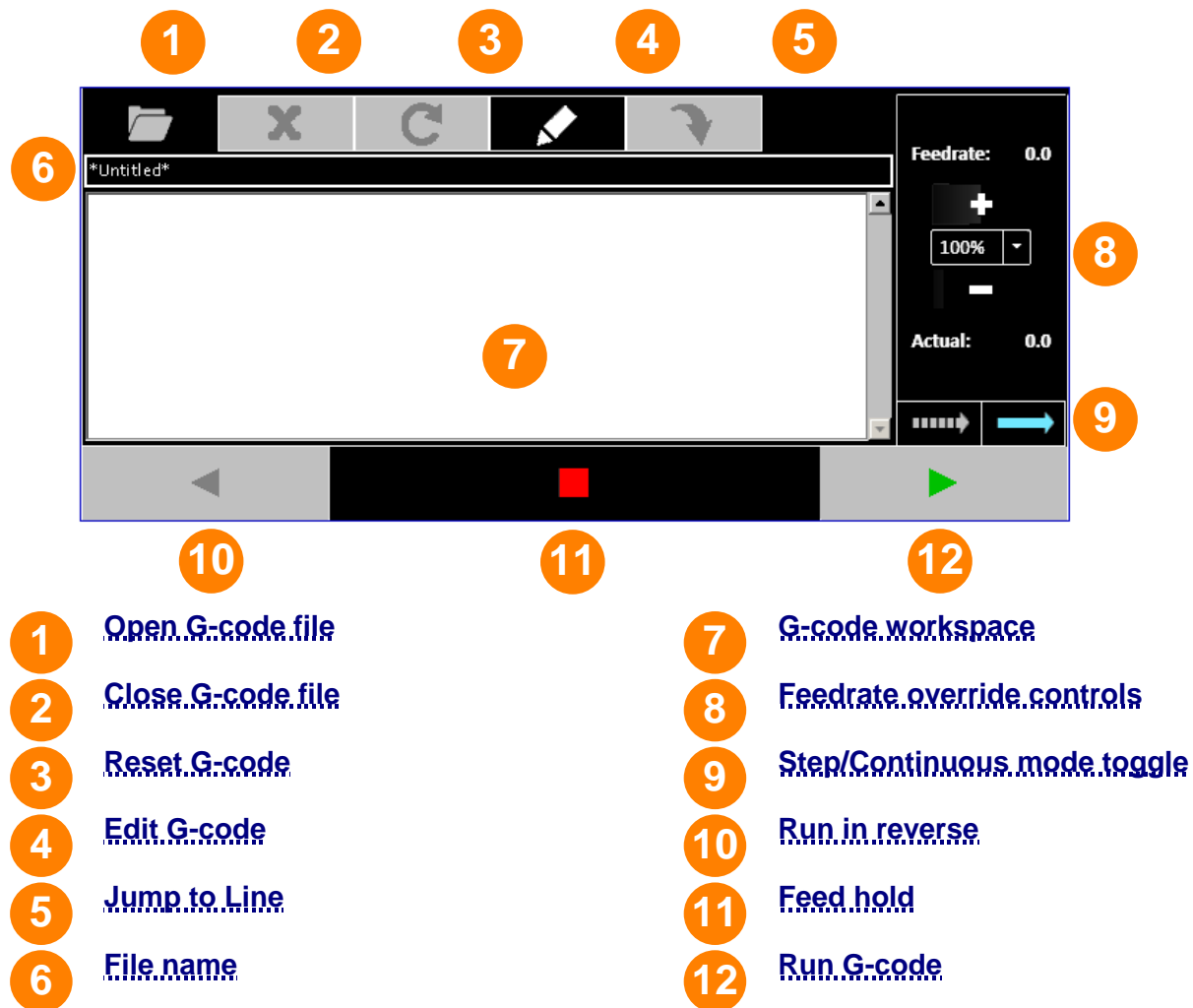
Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed.

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, an external 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:



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 **File name** field above the toolpath.

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.

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.

Edit G-code

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

Jump to Line

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

File name

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

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. One hundred percent override corresponds to zero change in feed rate. This function does not affect rapid move speeds, where feed rate is undefined.

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

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.

Pan and zoom tools



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



Pan

The pan tool allows you to click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window also allows you to pan without the use of the pan tool.



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.



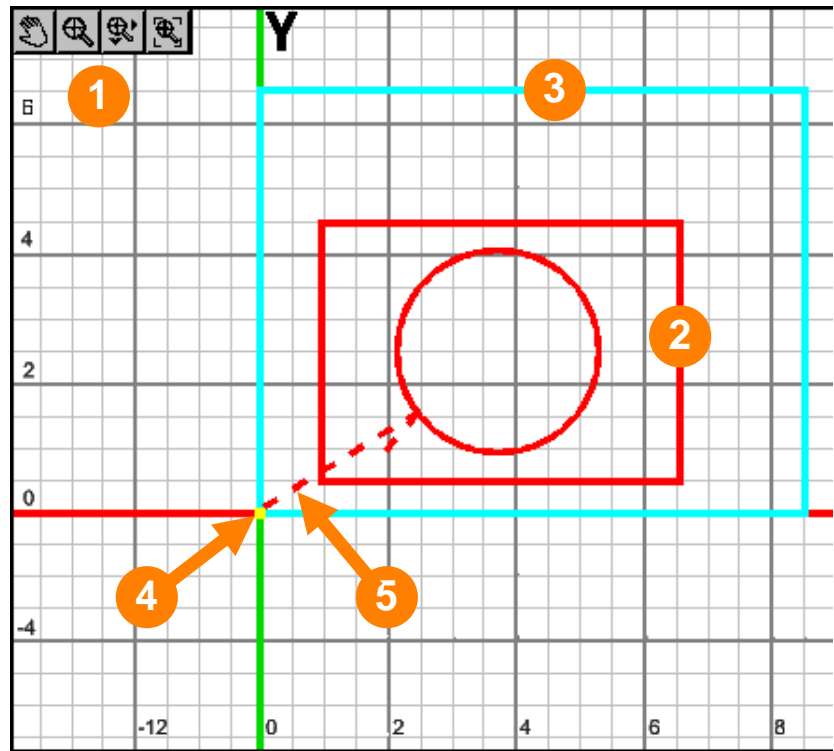
Zoom to Fit

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 **Configuration** screen.

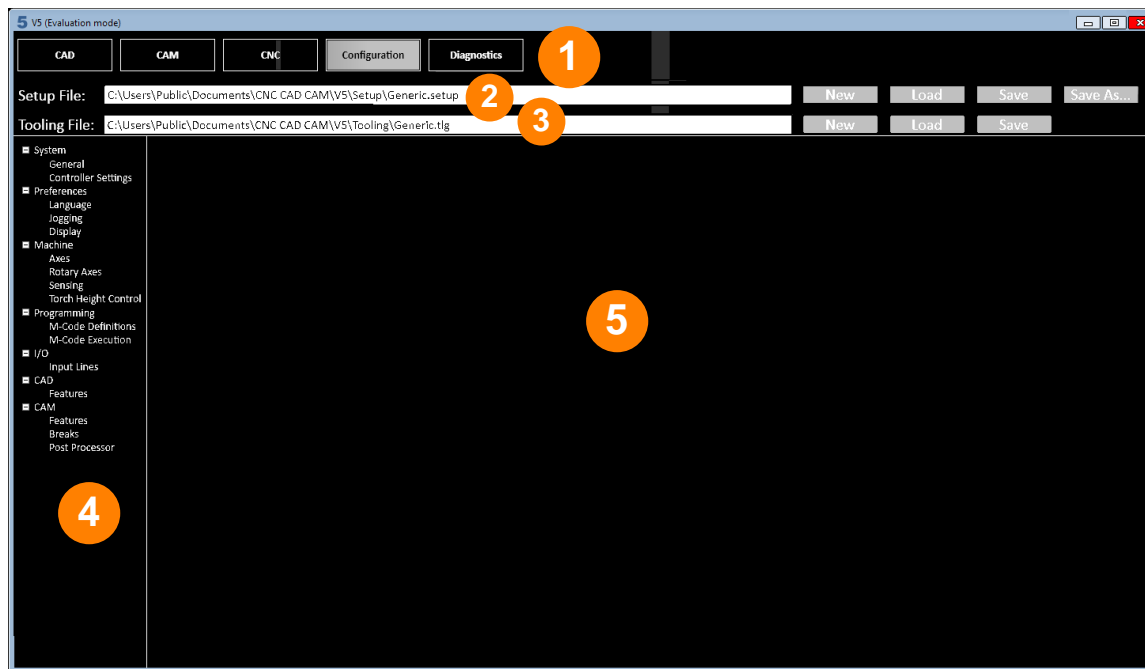
The workspace pictorially displays the G-code as it is written in the program editor.



- | | |
|--|--|
| <p>1 <u>Pan and zoom tools</u></p> <p>2 Solid red lines show the toolpath</p> <p>3 The maximum dimensions which a tool may not exceed are represented by a thick blue box. 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. Click the machine button to revise the feature and return to FlashCut CNC.</p> | <p>4 The current tool position is shown by a yellow dot.</p> <p>5 Dotted red lines represent rapid moves</p> |
|--|--|

Configuration

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. Click the **Configuration** tab to modify FlashCut settings.



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

- | | |
|---|--|
| 1 <u>Tab</u> | 4 <u>Menu section</u> |
| 2 <u>Setup file (*.STP extension)</u> | 5 <u>Configuration workspace</u> |
| 3 <u>Tooling file (*.TLG extension)</u> | |

Tabs

Use the tabs at the top of the screen to choose the type of tasks you'd like to do.



- CAD displays [FlashCut.CAD](#)
- CAM displays [FlashCut.CAM](#)
- CNC displays [FlashCut.CNC](#)
- Configuration displays the FlashCut [Configuration](#) interface
- Diagnostics displays the FlashCut [Diagnostics](#) interface

Setup file (*.STP extension)

When FlashCut is first loaded, it prompts you to select a setup file. 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 an image of each list. Clicking the triangle 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 STP file format.



The **New** button allows you to duplicate a previously-created STP 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 5\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 STP file and destination.

Tooling file (*.TLG extension)

When FlashCut is first loaded, it prompts you to select a tooling file. If you don't have a tooling file to load, click **Cancel** to tell the program not to open any tooling files.



Each tooling file (*.tooling, *.tlg) contains settings for up to 100 tools. Each tool has an associated tool number, description, diameter and XYZ offset. The diameter is used by the cutter compensation commands (G41 and G42). The XYZ offset is used by the tool length/geometry compensation commands (G43 and G44).

Menu section

The menu section of the configuration main screen presents the following available configuration option types:

- 1 System
- 2 Preferences
- 3 Machine
- 4 Programming
- 5 I/O
- 6 CAD
- 7 CAM

- ▣ System
 - 1 General
 - Controller Settings
- ▣ Preferences
 - 2 Language
 - Jogging
 - Display
- ▣ Machine
 - 3 Axes
 - Rotary Axes
 - Sensing
 - Torch Height Control
- ▣ Programming
 - 4 M-Code Definitions
 - M-Code Execution
- ▣ I/O
 - 5 Input Lines
- ▣ CAD
 - 6 Features
- ▣ CAM
 - 7 Features
 - Breaks
 - Post Processor

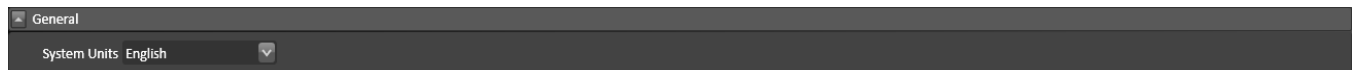
System

The **System** configuration option type presents two configuration options:

- **General**
- **Controller Settings**

General

The **General** configuration option presents the **System Units** controls.



System Units

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

- English
- Metric

Machine Information

Machine information is where you set FlashCut for the specific type of machine being controlled.

Machine Type

Specify the type of machine tool that the software is running. Plasma/Oxyfuel is the only machine type currently supported. Future releases of the software will support various other machine tools.

Use Oxyfuel Cutting

This specifies that an Oxyfuel torch is being used. The sequence of events is different for Oxyfuel cutting, albeit similar to plasma, so this enables settings that accommodate these differences.

Controller Settings

Controller Settings displays the following settings sections:

- [Controller](#)
- [Motor Signals](#)
- [Signal State Preferences](#)

Controller

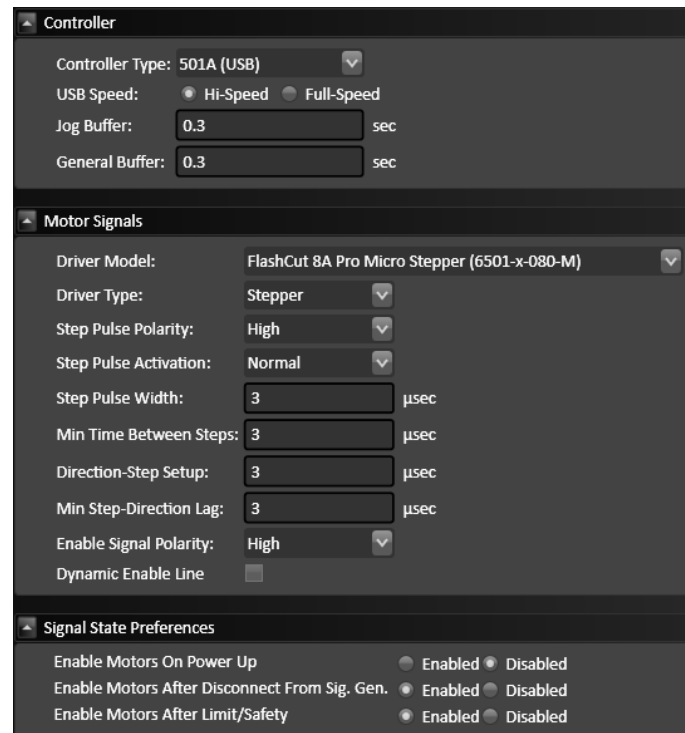
The **Controller** settings section presents the following controls:

- [Controller Type](#)
- [USB Speed](#)
- [Jog Buffer \(sec\)](#)
- [General Buffer \(sec\)](#)

Controller Type

Specifies the type of signal generator (controller) you are using. The two available FlashCut models are:

- 401A (serial port connection)
- 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.

Motor Signals

The **Motor Signals** settings section presents the following controls:

- **Driver Model**
- **Driver Type**
- **Step Pulse Polarity**
- **Step Pulse Activation**
- **Step Pulse Width (usec)**
- **Min Time Between Steps (usec)**
- **Direction-Step Setup (usec)**
- **Min Step Direction Lag (usec)**
- **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).

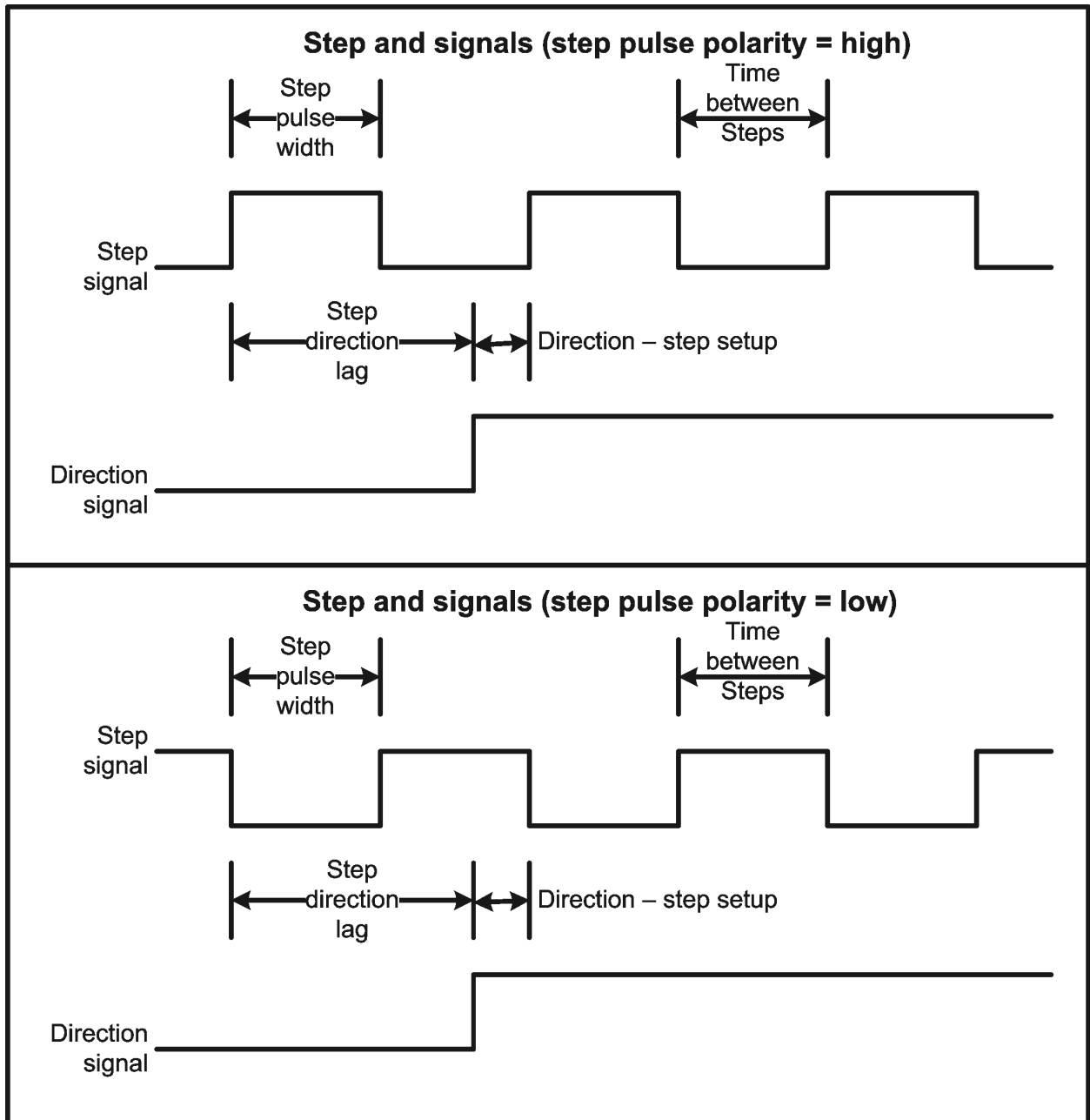
If you are using non-FlashCut drivers, choose **Other**. This enables all fields, allowing you to enter the proper settings based on your motor driver requirements. Note that some signal fields do not apply to all signal generator models, and may be disabled based on your controller settings (signal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

The **Signal State Preferences** settings section presents the following controls:

- **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

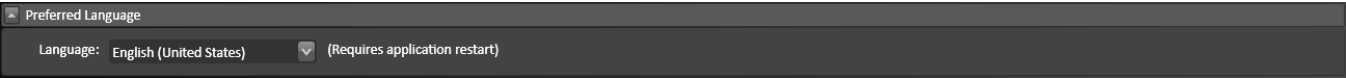
Sets the state to which the signal generator drives the motor enable line after a limit or safety error. See [Line Setup](#) for more information on these errors.

Preferences

The **Preferences** configuration option type presents three configuration options:

- **Languages**
- **Jogging**
- **Display**

Languages



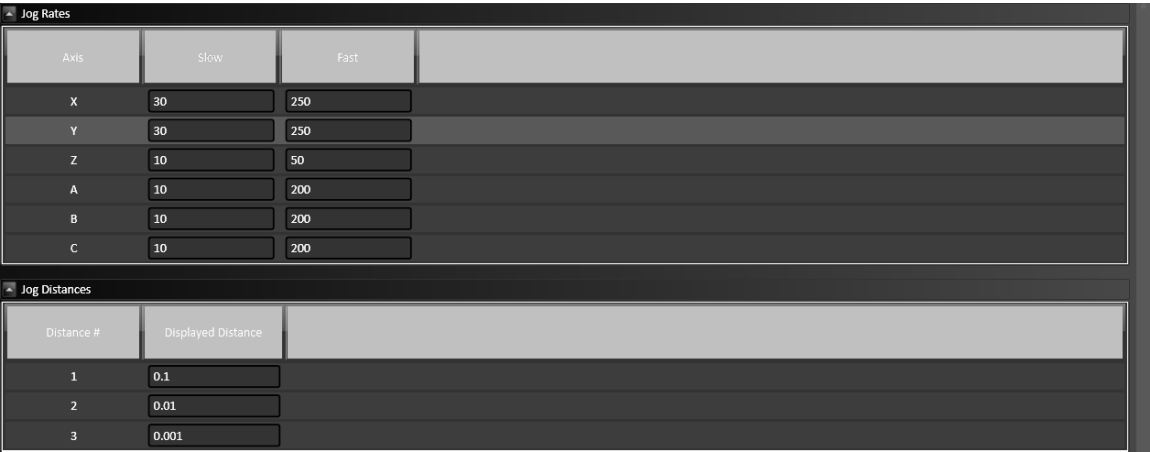
Use the **Languages** configuration option to choose the language FlashCut uses for its user interface. You may choose from:

- English (United States)
- Spanish (Spain)
- Italian (Italy)
- German (Germany)
- French (France)
- Portuguese (Portugal)
- Russian (Russia)

Jogging

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

- [Jog Rates](#)
- [Jog Distances](#)



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 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.

Jog Distances	
Distance #	Displayed Distance
1	0.1
2	0.01
3	0.001

Display

Use the **Display** options to adjust the settings for axis display preferences. Check the appropriate check box for the desired axis settings.

Axis Display	
Display X axis horizontally in the viewport (and Y vertically):	<input checked="" type="checkbox"/>
If X is horizontal, positive is to the right.	<input checked="" type="checkbox"/>
If X is vertical, positive is up:	<input type="checkbox"/>



Machine

The **Machine** configuration option type presents four configuration options:

- Axes
- Rotary Axes
- Sensing
- Torch Height Control

Axes

The axes option presents these parameters:

- [Motor Line/Axis Mapping](#)
- [Axis Settings](#)
- [Axis Drive Parameters](#)
- [Feedrates and Ramping](#)

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.

Motor Line/Axis Mapping					
Line #	Axis		Sub Axis		Direction
1	X	▼	N/A	▼	<input checked="" type="radio"/> Positive <input type="radio"/> Negative
2	Y	▼	N/A	▼	<input checked="" type="radio"/> Positive <input type="radio"/> Negative
3	Z	▼	N/A	▼	<input checked="" type="radio"/> Positive <input type="radio"/> Negative
4	A	▼	N/A	▼	<input checked="" type="radio"/> Positive <input type="radio"/> Negative
5	B	▼	N/A	▼	<input checked="" type="radio"/> Positive <input type="radio"/> Negative

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, B, C) 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.

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. See [Line Setup](#) later in this section for more information.

Direction

Sets the direction of rotation for the motor connected to each signal generator axis. Depending on how a motor is wired, the same signal from the motor driver can turn it clockwise or counter clockwise. Use the jog buttons to make sure that a positive move in each axis on the screen corresponds to a positive move in each axis on the machine tool. Note that the direction of movement is defined as the direction of the tool relative to the workpiece. For example, on a typical milling machine, a positive X move (tool movement to the right) requires table movement to the left. If any direction is incorrect, change the Motor Direction from Positive to Negative (or vice-versa) to reverse the correspondence between the software and machine tool.

Axis Settings

Axis	Use Axis	Limited travel	Length
X	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	40
Y	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	40
Z	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	15
A	<input type="checkbox"/>	<input type="checkbox"/>	360
B	<input type="checkbox"/>	<input type="checkbox"/>	360
C	<input type="checkbox"/>	<input type="checkbox"/>	360

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.

Axis Drive Parameters

The axis drive parameters are shown in two columns.

Axis

Each letter corresponds to a specific axis that is expected to be controlled from a G-Code program.

Tool Positioning Resolution (in/step)

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.

Axis Drive Parameters	
Axis	Tool Positioning Resolution (in/step)
X	0.000250000000
Y	0.000250000000
Z	0.000250000000
A	0.090000000000
B	0.090000000000
C	0.090000000000

Feedrates and Ramping

Feedrates and Ramping					
Axis	Start/Stop Feedrate (in/min)	Maximum Feedrate (in/min)	General Ramp Rate (in/sec/sec)	Feedrate Move Ramp Rate (in/sec/sec)	Continuous Contouring Feedrate (in/sec)
X	10	200	2	2	10
Y	10	200	2	2	10
Z	10	200	2	2	10
A	360	1440	125	125	360
B	360	1440	125	125	360
C	360	1440	125	125	360

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:

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

Note 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.)

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

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.

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.

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

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.

Choose Feedrate/Ramping from the Configuration menu.

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

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.

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

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

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.

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.

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

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.

Choose Feedrate/Ramping from the Configuration menu.

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

Repeat this process for all axes.

General Ramp Rate (in/sec/sec)

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

Ramping Rates typically range from 1 to 10 in/sec². Slower ramping rates require more time to ramp up to the maximum feedrate and to ramp down to a stop. This may become a potentially dangerous situation when using the Feed Hold button or jogging since the machine will take longer to come to a complete stop. The goal is to choose a fast ramping rate that will start and stop the tool responsively without losing steps. Fast ramping rates can also allow acceleration past resonant speeds of a stepper motor.

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

- a. Enter 1 in/sec² in the General Ramping Rate text box for the X axis. (This is a conservative ramping rate.)
- b. Choose OK.
- c. Choose the Point button on the Control Selection Box. Select Any Point in the Point List and Incremental from the Coordinate System pull-down menu. Enter 1.0 (or 25 if in metric mode) in the X text box. Choose Rapid from the Rate Mode pull-down menu. Make sure you have room to move the X axis 1 inch, and then choose Move. If the table doesn't reach a continuous speed before ramping down to a stop, there isn't enough room to fully ramp up and down, and you should increase the length of the move.
- d. If the motor slips, repeat this process with a lower ramping rate. If the motor doesn't slip, try a higher ramping rate. Note that slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if your machine has them.
- e. Repeat the above steps until you determine an optimal General Ramping Rate for the X axis.
- f. Choose Feedrate/Ramping from the Configuration menu.
- g. Enter 70% of the highest no-slip ramping rate you found in the X axis General Ramping Rate text box.
- h. Copy the General Ramping Rate to the Feedrate Move Ramping Rate text box. If you determine that your system would benefit from a lower ramping rate for feedrate (cutting) moves, enter a lower Feedrate Move Ramping Rate.
- i. Repeat this process for all axes.

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

Feedrate Move Ramp Rate (in/sec/sec)

Sets the rate of acceleration and deceleration for feedrate moves.

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.

Rotary Axes

If the system is using a rotary axis, it must be configured here.

Rotary Axis Arrangement

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 Configuration

Click the arrow next to Rotary Axis Configuration to display a drop-down list box. You may choose from the following configuration options:

- None
- Other Configuration
- Rotary Table (Four Axes)
- Dual Rotary Table (Five Axes)
- Rotary Knife

General

These sequences simplify the process of defining program coordinates, tool offsets and part geometry. Define sensing operation parameters here.

F Command for Rotary Moves

Select whether the F command for the rotary axis is programmed using a linear feedrate (distance/minute) or a rotary feedrate (degrees/minute).

- Linear Feedrate (Distance/Minute)
- Rotary Feedrate (Distance/Minute)

Sensing

These sequences simplify the process of defining program coordinates, tool offsets and part geometry. Define sensing operation parameters here.

Program Zero Sensing

You can use program zero sensing to detect program zero for any axis.

Use Program Zero Sensing

Activates this feature

Sensing Direction

The direction of the move towards the sensor.

Sensing Feedrate

The feedrate for the move towards the sensor.

Monitor Input Line

The input line for the program zero sensor.

Max Distance to Move

The maximum distance the axis will move to find the sensor.

Retract Distance

The distance the axis retracts after tripping the sensor

Sensor Offset

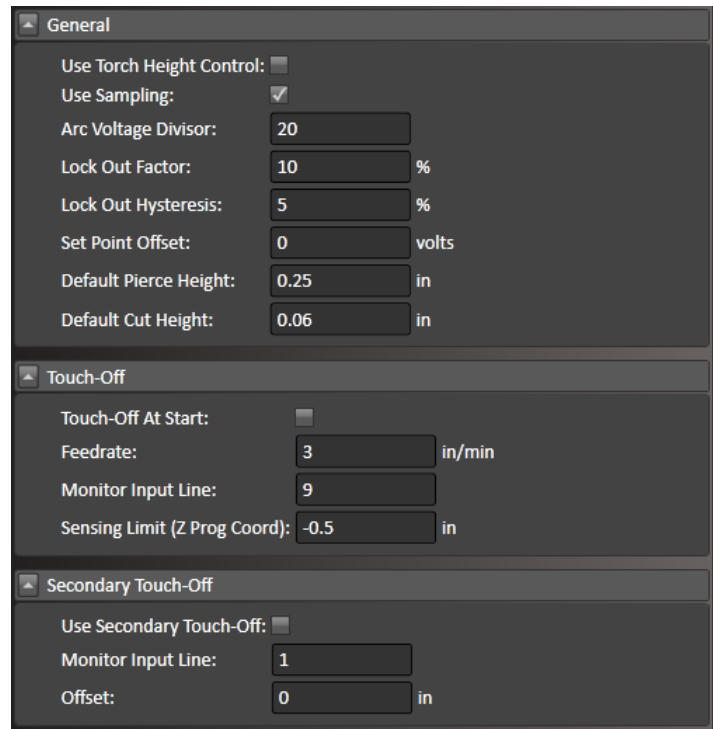
The thickness of the sensor

The screenshot shows the 'Program Zero Sensing' dialog box. It has a title bar with a minus, maximize, and close button. Inside, there is a checkbox labeled 'Use Program Zero Sensing' which is currently unchecked. Below this, there are several settings: 'Sensing Direction' with radio buttons for 'Positive' and 'Negative' (the 'Negative' button is selected); 'Sensing Feedrate' with a text input field containing '3' and a unit label 'in/min'; 'Monitor Input Line' with a dropdown menu showing '1'; 'Max Distance To Move' with a text input field containing '0.25' and a unit label 'in'; 'Retract Distance' with a text input field containing '0.1' and a unit label 'in'; and 'Sensor Offset' with a text input field containing '1' and a unit label 'in'.

Torch Height Control

Define these Torch Height Control parameters here.

- [General](#)
- [Touch-Off](#)
- [Secondary Touch-Off](#)



General

Use Torch Height Control: ☐

Use Sampling: ☒

Arc Voltage Divisor: 20

Lock Out Factor: 10 %

Lock Out Hysteresis: 5 %

Set Point Offset: 0 volts

Default Pierce Height: 0.25 in

Default Cut Height: 0.06 in

Touch-Off

Touch-Off At Start: ☐

Feedrate: 3 in/min

Monitor Input Line: 9

Sensing Limit (Z Prog Coord): -0.5 in

Secondary Touch-Off

Use Secondary Touch-Off: ☐

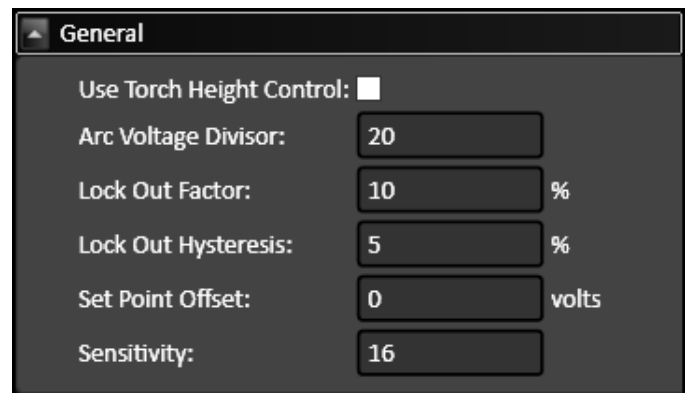
Monitor Input Line: 1

Offset: 0 in

General

Torch height control General settings contains the following options:

- [Use Torch Height Control](#)
- [Arc Voltage](#)
- [Lock Out Factor](#)
- [Lock Out Hysteresis](#)
- [Set Point Offset](#)
- [Sensitivity](#)



General

Use Torch Height Control: ☐

Arc Voltage Divisor: 20

Lock Out Factor: 10 %

Lock Out Hysteresis: 5 %

Set Point Offset: 0 volts

Sensitivity: 16

Use Torch Height Control

When this box is checked, then all of the torch height control features will be available on the CNC control panel. You will still have the ability to turn torch height control on and off at the CNC control panel.

Arc Voltage Divisor

This value should match the exact divisor that the plasma machine is set to. Note that in order for the torch height control system to work properly, **(maximum raw voltage) / (arc voltage divisor)** must be smaller than 6V.

Example 1

If the maximum raw voltage for a given plasma machine is 300V, then the divisor must be at least 50. Also note that the voltage isolation box has a gain of 2.0. So the actual divisor that you should put in the software currently is the divisor / 2.

Example 2

For example, if the divisor on your plasma machine is set to 50, then the arc voltage divisor in the software would be set to 25.

Note This will be changed so that you don't have to deal with our 2X gain. You will just put in the divisor that you have on your plasma machine and FlashCut automatically divides it by 2. In this example, you would set the arc voltage divisor to 50. Once it is revised to use this approach, the values in old setup files will be automatically adjusted.

Lock Out Factor

This factor is used to turn the THC off when the XY velocity of the torch is below a percentage of the programmed feedrate. This will prohibit the torch from diving into the workpiece at corners or anywhere else that the XY velocity slows down. For example if the Lock Out Factor is set to 90% and the program feedrate is F=200, then the THC will automatically turn off when the XY velocity drops below 180.

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). For example, if the Lock Out Hysteresis is 5%, then the THC will turn back on when the XY velocity goes above $(90\% + 5\%) * 200 = 190$.

Set Point Offset

Many times the CNC Control panel will read and display a small voltage (on the order of 0-10V) when the torch is off. If you put this voltage value in the Set Point Offset box, it will bring this value back to zero on the display.

Sensitivity

This is a factor that 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. If there is excess up and down movement in the torch, then this number should go up in value. If it is not responsive enough, then this number should go down in value. (In a later release, this number should be a voltage – from 0.05V to 2V in increments of 0.05V.) Where 0.05V is the most sensitive.

The equation is presently:

$$((\text{Value sent by PC} + 1) * 128 * 0.003223 * 50) / 690 = \text{BIN1 voltage}$$

The BIN1 voltage is the +/- range in which no adjustment is made.

For example, the PC sends down a 15 which makes the BIN1 voltage = 0.478312 which means that no adjustment will be made until the arc voltage measurement is more than +/-0.478312 off.

$$\text{BIN2} = 2 * \text{BIN1} \text{ and } \text{BIN3} = 3 * \text{BIN1} \text{ and } \text{BIN4} = 4 * \text{BIN1}$$

BIN1 uses the slowest adjustment rate. BIN2 uses a medium slow rate, BIN3 uses a medium fast rate and anything farther off than BIN4 adjusts at the fastest rate.

Touch-Off

Torch height control Touch-Off settings contains the following options:

- **Touch Off At Start**
- **Feedrate**
- **Monitor Input Line**
- **Offset**
- **Sensing Limit (Z Prog Coord)**

Touch-Off At Start:	<input type="checkbox"/>
Feedrate:	3 in/min
Monitor Input Line:	9
Offset:	0 in
Sensing Limit (Z Prog Coord):	-0.5 in

Touch Off At Start

Note that this is the primary switch that the touch off sensing routine will seek. If it fails for some reason (for example - rust, paint, etc prohibiting good electrical contact on an ohmic sensor), then it will either reach its sensing limit, or it will hit the secondary touch off if activated.

Feedrate

This is the feedrate in which the torch will move down to sense the workpiece surface.

Monitor Input Line

This is the input line that the touch sensor is connected to. Typically this is the ohmic sensor signal which is mapped to input line 19.

Offset

This is extra distance that the Z axis travels in the upwards direction after the secondary switch senses the workpiece surface. This distance can account for any hysteresis in the sensing switch or any slop or backlash in the mechanical system that raises and lowers the torch.

Sensing Limit (Z Prog Coord)

This is the minimum Z axis program coordinate that the torch can travel to without sensing the workpiece surface. If it travels to the sensing limit before it finds the surface, motion will stop and an error will appear saying that the sensing limit has been reached.

Secondary Touch-Off

In some cases it may be desirable to use two different techniques for the touch off sequence, this is where the secondary approach is defined. The secondary touch off can also act as a safety mechanism in case there is a failure in the primary touch off method.

Torch height control Secondary Touch-Off settings contains the following options:

- **Use Secondary Touch-Off**
- **Monitor Input Line**
- **Offset**

Use Secondary Touch-Off:	<input type="checkbox"/>
Monitor Input Line:	1
Offset:	0 in

Use Secondary Touch-Off

When this box is checked, the system will look for a backup switch to trip during the touch off process in case the primary touch off method fails. Typical uses for this is if the primary switch is an ohmic sensor and periodically rust, paint, dirt or some other coating prohibits the ohmic sensor to make good electrical contact with the workpiece surface.

Monitor Input Line

This is the input line that the secondary touch sensor is connected to. Typically this is the mechanical sensor signal which can be mapped to any open input line.

Offset

This is extra distance that the Z axis travels in the upwards direction after the secondary switch senses the workpiece surface. This distance can account for any hysteresis in the sensing switch or any slop or backlash in the mechanical system that raises and lowers the torch.

Programming

The **Programming** configuration option type presents two configuration options:

- [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.

MCode

The number for the M-code that you want to define.

MCode	Description	Execute Before Move	Execute Delay First	Delay (sec)	Output Lines
3	Spindle On	<input checked="" type="checkbox"/>	<input type="checkbox"/>	1	2 <input type="button" value="v"/> <input type="button" value="On"/> <input type="button" value="x"/>
5	Spindle Off	<input type="checkbox"/>	<input type="checkbox"/>	0	2 <input type="button" value="v"/> <input type="button" value="Off"/> <input type="button" value="x"/>

Description

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

Execute Before Move

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

Execute Delay First

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

Delay (sec)

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

Output Lines

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

Add M-Code Button

Adds a new line to define an M-code.

Delete M-Code Button

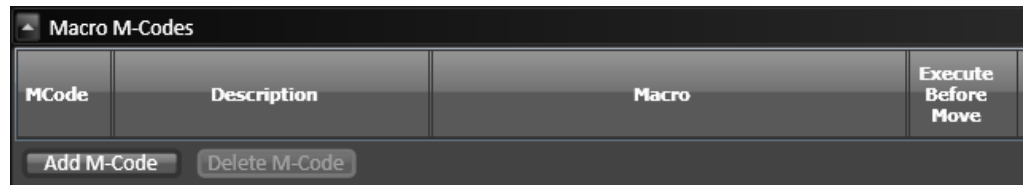
Deletes the highlighted M-code definition.

Macro M-Codes

Flashcut can also use M-codes to call a macro.

MCode

The number for the M-code that you want to define.

**Description**

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

Macro

Click in this field, 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:

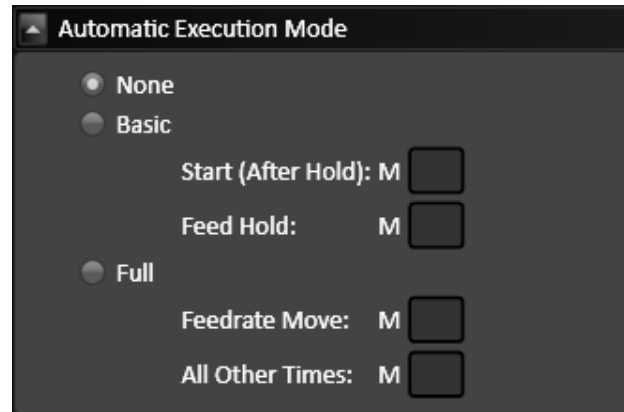
- Automatic Execution Mode
- Error! Reference source not found.
- Execute After Motion Errors
- Execute at Start/End of Motion
- Execute During Ramping
- Execute While Connecting/Disconnecting

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.



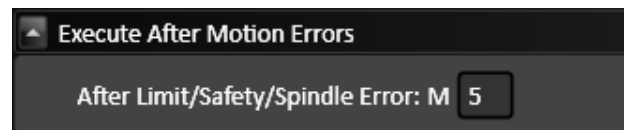
- **Start (After Hold)** FlashCut can execute an M-code whenever **Start** is clicked after a feed hold has occurred (but not when initially starting the file).
- **Feed Hold** FlashCut can execute an M-code whenever **Feed Hold** is clicked, or an error occurs.

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).

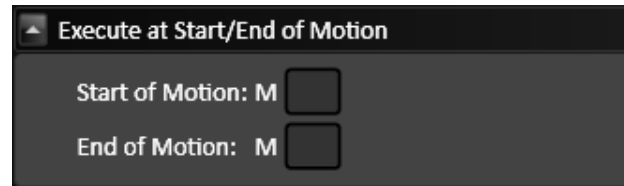
When running a lathe, and especially when cutting threads, this mode is highly recommended. If an error occurs (e.g. a limit switch gets tripped), FlashCut immediately stops moving the tool. The spindle must then be stopped, or else there will be interference between the stationary tool and the spinning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

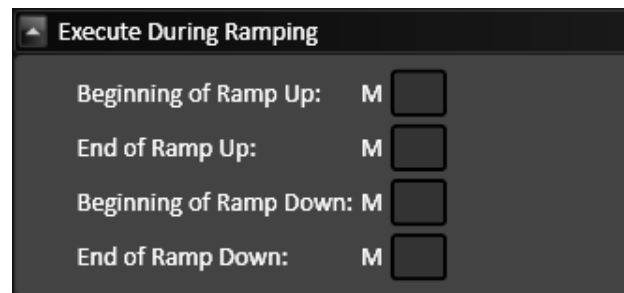
Note Motion includes actual machine moves as well as dwells.



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.

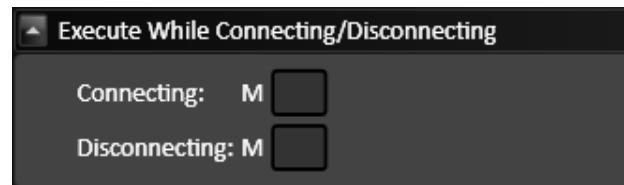
- **Beginning of Ramp Up** – FlashCut executes this M-code when ramping up begins.
- **End of Ramp Up** – FlashCut executes this M-code when ramping up ends.
- **Beginning of Ramp Down** – FlashCut executes this M-code when ramping down begins.
- **End of Ramp Down** – FlashCut executes this M-code when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

The **I/O** configuration option type presents two configuration options:

- **General**
- **Line Setup**

General

These parameters definitions are system wide.

M100/M101 debounce is minimum time that required for the system to recognize a state change.

Line Setup

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

General

M100/M101 Debounce: 0.25 sec

Line Setup

Set All Wiring: Normally Open Normally Closed

Line #	Description	Function	Axis	Sub Axis	Position	Wiring
1		Unused	N/A	N/A	+ -	N.O. N.C.
2		Unused	N/A	N/A	+ -	N.O. N.C.
3		Unused	N/A	N/A	+ -	N.O. N.C.
4		Unused	N/A	N/A	+ -	N.O. N.C.
5		Unused	N/A	N/A	+ -	N.O. N.C.
6		Unused	N/A	N/A	+ -	N.O. N.C.
7		Unused	N/A	N/A	+ -	N.O. N.C.
8		Unused	N/A	N/A	+ -	N.O. N.C.
9		Unused	N/A	N/A	+ -	N.O. N.C.
10		Unused	N/A	N/A	+ -	N.O. N.C.
11		Unused	N/A	N/A	+ -	N.O. N.C.
12		Unused	N/A	N/A	+ -	N.O. N.C.
13		Unused	N/A	N/A	+ -	N.O. N.C.
14		Unused	N/A	N/A	+ -	N.O. N.C.
15		Unused	N/A	N/A	+ -	N.O. N.C.
16		Unused	N/A	N/A	+ -	N.O. N.C.
17		Unused	N/A	N/A	+ -	N.O. N.C.
18		Unused	N/A	N/A	+ -	N.O. N.C.
19		Unused	N/A	N/A	+ -	N.O. N.C.
20		Unused	N/A	N/A	+ -	N.O. N.C.
21		Unused	N/A	N/A	+ -	N.O. N.C.
22		Unused	N/A	N/A	+ -	N.O. N.C.
23		Unused	N/A	N/A	+ -	N.O. N.C.
24		Unused	N/A	N/A	+ -	N.O. N.C.
25		Unused	N/A	N/A	+ -	N.O. N.C.
26		Unused	N/A	N/A	+ -	N.O. N.C.
27		Unused	N/A	N/A	+ -	N.O. N.C.
28		Unused	N/A	N/A	+ -	N.O. N.C.
29		Unused	N/A	N/A	+ -	N.O. N.C.
30		Unused	N/A	N/A	+ -	N.O. N.C.
31		Unused	N/A	N/A	+ -	N.O. N.C.
32		Unused	N/A	N/A	+ -	N.O. N.C.

You may set the following options for each input line:

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

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

Line

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

Description

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

Function

The function of the switch as follows:

- **Home/Limit** – The switch will be used for homing and limiting travel.
- **Home** – The switch will be used for homing only.
- **Limit** – The switch will be used for limiting travel only.
- **Safety** – The switch will stop motion immediately (e.g. a servo error line.) The switch behaves the same as a Limit switch, but does not have a Location associated with it.
- **Control** – The switch will be used for general control purposes (e.g. for tool length sensing or program zero sensing).
- **Feed Hold** – The switch will cause a feed hold to occur, just as if the Feed Hold button were clicked.
- **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 subaxes 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.

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

The **CAD** configuration option type presents one configuration option:

- **Features**

Features

The Features option presents one configuration option:

- **Select Features to Display**

Select Features to Display

- **Import Silhouette Image**
- **Import Centerline Image**

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

The **CAM** configuration option type presents three configuration options:

- **Features**
- **Breaks**
- **Post Processor**

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 CAD module.

Breaks

This section presents the following topics:

- [Part Start Point](#)
- [Cutout Start Point](#)

Part Start Point

This section defines the default values for the break placements and lead definitions **for parts**. The parameters are identical to the section below titled [Cutout Start Point](#) 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

Cutout Start Point

This section defines the default values for the break placements and lead definitions **for cutouts**. This is identical to the **Part Start Point** 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

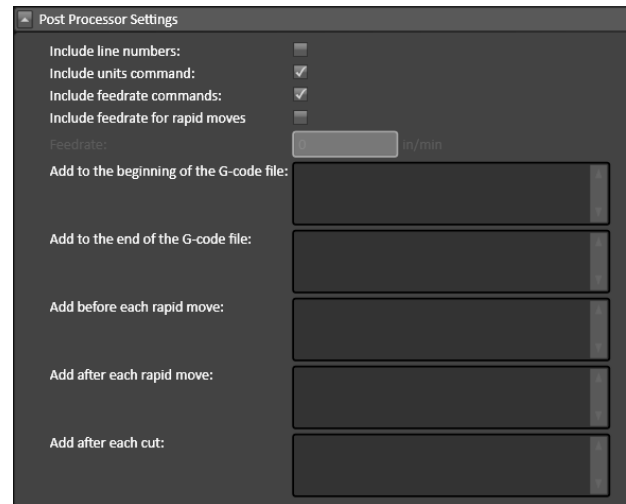
There are two types of post processor settings:

- [Post Processor Settings](#)
- [Save G-Code to FTP Server](#)

Post Processor Settings

Use the post processor settings to automatically add these types of code to your G-code file:

- [Include line numbers](#)
- [Include units command](#)
- [Include federate commands](#)
- [Include feedrate for rapid moves](#)
- [Add to the beginning of the G-code file](#)
- [Add to the end of the G-code file](#)
- [Add before each rapid move](#)
- [Add after each rapid move](#)
- [Add after each cut](#)



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 federate 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.

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.

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).

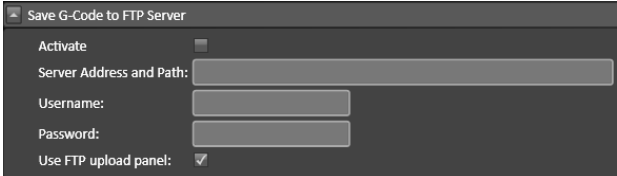
Add after each cut

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. There are five settings:

- **Activate**
- **Server Address and Path**
- **Username**
- **Password**
- **Use FTP upload panel**

**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.

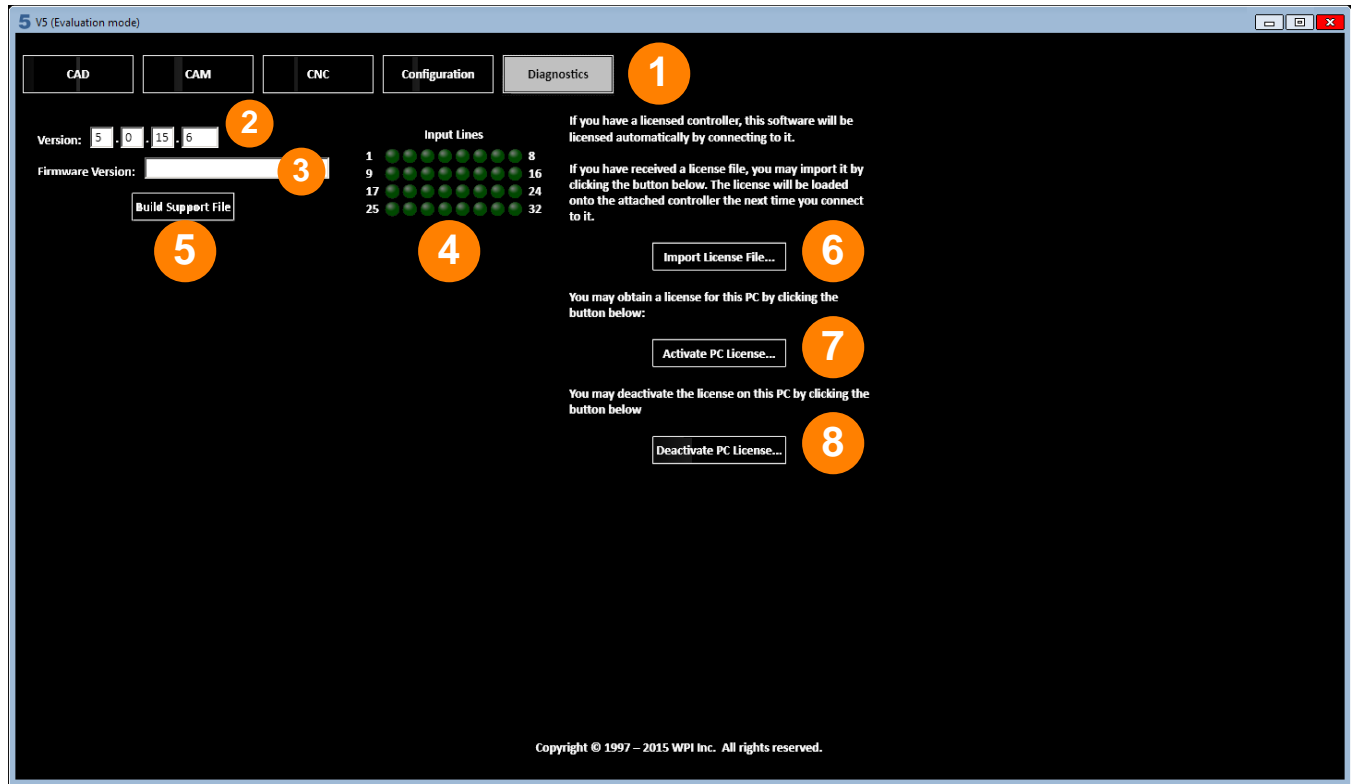
If **Use FTP upload panel** is unchecked: the file is uploaded with a default filename (eg, gcode.nc).

Configuration workspace

The configuration workspace is where you see all the options for each configuration type.

Diagnostics

Click the **Diagnostics** tab to display the FlashCut diagnostics screen shown here.



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

- | | | | |
|---|----------------------------|---|--|
| 1 | Tabs | 5 | Build Support File button |
| 2 | Version | 6 | Import License File... button |
| 3 | Firmware version | 7 | Activate PC License... button |
| 4 | Input Lines display | 8 | Deactivate PC License... button |

Tabs

Use the tabs at the top of the screen to choose the type of tasks you'd like to do.



- CAD displays [FlashCut.CAD](#)
- CAM displays [FlashCut.CAM](#)
- CNC displays [FlashCut.CNC](#)
- Configuration displays the FlashCut [Configuration](#) interface
- Diagnostics displays the FlashCut [Diagnostics](#) interface

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.

Build Support File button

Builds a compressed ZIP file that contains all the information about your system that's needed by FlashCut technical support to answer a support question. FlashCut displays the file Save As dialog so you can save the support file anywhere that's convenient for attachment to an email (such as the Windows desktop.) The default location is **C:\FlashCut Data\Support**.

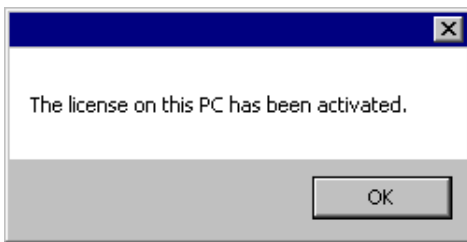
Input Lines display

Shows the current status of the input lines. Hover over the indicator with the cursor to see the description of each particular input line. The indicator is illuminated when the input line is tripped (regardless of the whether the line is configured as normally open or normally closed) and off when the input line is not tripped.

Import License File... button

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.

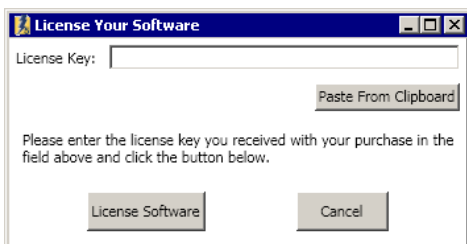


6. Click **OK**. The license will be loaded into your controller the next time you connect to it.

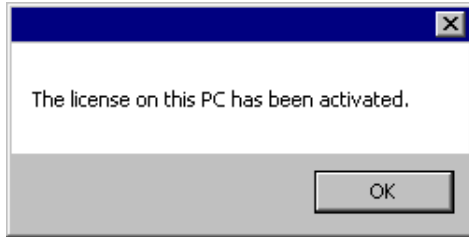
Activate PC License... button

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 this notice to confirm that the license has been activated. The license will be loaded into your controller the next time you connect to it.

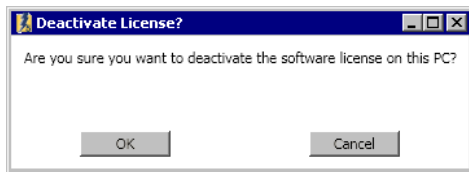


6. The license will be loaded into your controller the next time you connect to it.

Deactivate PC License... button

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.