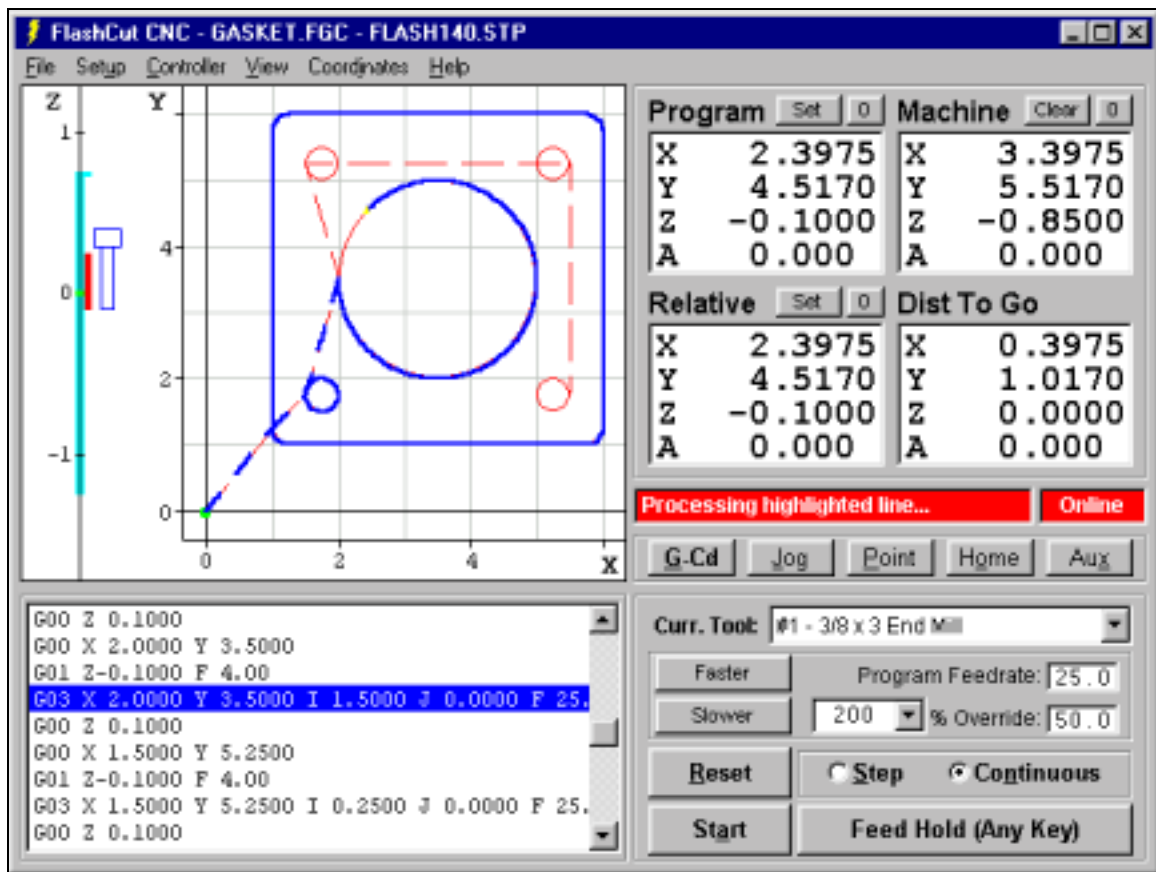


FLASHCUT CNC

TM

Computer Numerical Control for Windows
Version 1.4

User's Guide



© 1997-1999 WPI, Inc. All rights reserved.

West Coast Office
1263 El Camino Real
Menlo Park, CA 94025
Phone (650) 853-1444 ♦ Fax (650) 853-1405
www.flashcutcnc.com

East Coast Office
4949 St. Elmo Avenue, Suite 225
Bethesda, MD 20814
Phone (301) 718-7300 ♦ Fax (301) 718-7301
www.flashcutcnc.com

Table of Contents

GETTING STARTED	1
THANK YOU	1
PRODUCT SUPPORT	1
NEW FEATURES	1
SYSTEM REQUIREMENTS	2
INSTALLING FLASHCUT CNC	2
USING THE MOUSE	3
CHOOSING COMMANDS	3
<i>Choosing a Command by Using the Mouse</i>	3
<i>Choosing a Command by Using the Keyboard</i>	4
<i>Using Shortcut Keys</i>	4
<i>Using the TAB and ARROW Keys</i>	4
USING STANDARD WINDOWS CONTROLS	4
<i>Radio Buttons</i>	4
<i>Pull-Down Menus</i>	4
<i>Text Boxes</i>	4
<i>Command Buttons</i>	5
SAFETY AND USAGE GUIDELINES	6
ABOUT THIS MANUAL	7
MAIN SCREEN FEATURES	9
PULL-DOWN MENU BAR	9
<i>File Menu</i>	9
<i>Setup Menu</i>	10
<i>Controller Menu</i>	11
<i>View Menu</i>	12
<i>Coordinates Menu</i>	13
<i>Help Menu</i>	14
TOOL POSITION BOX	14
<i>Coordinate Systems</i>	14
<i>Command Buttons</i>	15
TOOLPATH VIEW PORT	15
CONTROL SELECTION BOX	17
<i>G-Code Control Panel</i>	17
<i>Jog Control Panel</i>	18
<i>Point Control Panel</i>	19
<i>Home Control Panel</i>	21
<i>Auxiliary Control Panel</i>	22
MESSAGE BOX	22
PROGRAM LISTING BOX	23
INITIAL SETUP	25
WINDOWS SETUP	25
SOFTWARE SETUP	25
<i>The Setup File</i>	25
<i>System Settings</i>	26
<i>Motor Signal Settings</i>	28
<i>Machine Tool Settings</i>	30
<i>Feedrate and Ramping Settings</i>	32
<i>Setting Up Backlash Compensation</i>	38
<i>Tooling Settings</i>	38
<i>Input Line Settings</i>	40
<i>Output Line Settings</i>	41
<i>G and M Code Settings</i>	43
<i>Message Settings</i>	43

<i>Setting Machine Zero</i>	44
SYSTEM PROGRAMMING	47
OPENING A G-CODE PROGRAM	47
IMPORTING A DXF FILE	47
USING THE PROGRAM EDITOR	50
G CODES SUPPORTED	52
M CODES SUPPORTED	53
OTHER COMMANDS SUPPORTED	53
KEY PROGRAMMING CONCEPTS	53
<i>Mode</i>	53
<i>Absolute vs. Incremental</i>	55
G AND M CODE REFERENCE	55
<i>G00 Rapid Tool Positioning</i>	55
<i>G01 Linear Interpolated Feedrate Move</i>	56
<i>G02 Clockwise Circular Feedrate Move</i>	57
<i>G03 Counter Clockwise Circular Feedrate Move</i>	58
<i>G04 Dwell</i>	59
<i>G17, G18, G19 Arc Plane Selection</i>	59
<i>G20, G21 Inch Units and Metric Units</i>	60
<i>G28 Return to Reference Point</i>	60
<i>G29 Return from Reference Point</i>	61
<i>G43, G44, G49, M06 Tool Change and Tool Length Compensation Commands</i>	62
<i>G52 Local Coordinate System</i>	66
<i>G73, G80, G81, G83, G98, G99 Canned Cycle Commands</i>	67
<i>G90 Absolute Positioning Mode</i>	70
<i>G91 Incremental Positioning Mode</i>	71
<i>M00 Program Pause</i>	71
<i>M30 End of Program</i>	71
<i>M98, M99, M02 Subroutine Commands</i>	71
<i>M100, M101 Wait for Input Line</i>	74
<i>M03, M05, M07, M08, M09, MXX Miscellaneous Device Control</i>	75
<i>F Feedrate Command</i>	75
<i>Program Comments</i>	75
TUTORIAL	77
STARTING FLASHCUT CNC	77
<i>Windows 3.1 or 3.11</i>	77
<i>Windows 95, 98 or NT</i>	77
CONFIGURING FLASHCUT CNC	77
LOADING A G-CODE FILE	78
VIEWING THE TOOL PATH	79
ANIMATING THE G-CODE FILE	81
EDITING A G-CODE FILE	82
GOING ONLINE	83
USING THE JOG CONTROLS	83
SETTING MACHINE ZERO	84
USING THE POINT MOVE	85
SETTING PROGRAM ZERO ON THE MACHINE TOOL	86
TESTING THE PROGRAM ON THE MACHINE TOOL	87
CUTTING THE PART	88
EXITING THE PROGRAM	88
TURNING OFF THE ELECTRONICS	88
GLOSSARY	89

Section 1. Getting Started

Thank You

Thank you for purchasing FlashCut™ CNC, the intuitive, powerful CNC control system for Windows®. No system is easier to set up and use than FlashCut CNC. You'll appreciate its intuitive controls and real time graphics as you quickly and accurately cut parts on your machine tool.

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

Product Support

We are committed to full support of FlashCut CNC products. Please contact us by email, phone or fax as follows:

West Coast

Phone: (650) 853-1444 (9:00 AM-5:00 PM, PST, M-F)

Fax: (650) 853-1405

East Coast

Phone: (301) 718-7300 (9:00 AM-5:00 PM, EST, M-F)

Fax: (301) 718-7301

Internet

E-mail: support@flashcutcnc.com

Web Site: www.flashcutcnc.com

New Features

We are constantly adding new features and capabilities to FlashCut CNC. Some of the important new features in the last two major software releases are:

Version 1.3 (Spring 1999)

- Continuous Contouring
- Canned Cycles (G73, G81, G83)
- New Homing Options
- Medium Jog Rate
- Many New Configuration Settings

Version 1.4 (Summer 1999)

- 4th Axis Support (Rotary or Linear)
- Monitoring Input Lines
- Subroutine Libraries
- Looping (in M98 Command)
- Main Screen Layout Updated

System Requirements

- IBM PC or 100% compatible with a 100MHz or faster 80486DX (Pentium or comparable processor preferred).
- At least 16MB of RAM (at least 64MB preferred for large G-code files).
- Hard drive with at least 20MB of space available.
- 3.5 inch, 1.44 MB floppy disk drive.
- VGA, SVGA or compatible video monitor.
- A Microsoft-compatible mouse. Note that for best performance, the mouse should connect directly to the bus (the standard mouse plug on most systems), or through a mouse bus port rather than through a **serial port**.
- One available RS-232 serial port. If the port has a 25-pin connector, a 9-pin male to 25-pin female adapter will be required.
- Microsoft Windows 95, 98 or NT or Microsoft or PC DOS version 3.1 or later running Microsoft Windows version 3.1 or 3.11.

Installing FlashCut CNC

It's a good idea to make a working copy of the FlashCut CNC disks and put the originals away in a safe place, before installing the program. Then if the working copy is damaged or lost, you can easily replace it.

If you are using Windows 3.1 or 3.11:

1. Start Windows
2. Place Disk 1 into drive A or drive B.
3. In the Windows Program Manager, choose the Run command from the File menu. In the Command Line text box, type **a:setup** if you are using disk drive A or **b:setup** if you are using disk drive B. Choose the OK button and follow the on-screen instructions.
4. To run FlashCut CNC simply double-click on the FlashCut CNC icon in the FlashCut CNC program group.

If you are using Windows 95, 98 or NT:

1. Start Windows
2. Place Disk 1 into Drive A or Drive B.
3. From the Start menu, choose the Run option. In the Open text box, type **a:setup** if you are using disk drive A or **b:setup** if you are using disk drive B. Choose the OK button and follow the on-screen instructions.
4. To Run FlashCut CNC, choose the Programs option from the Start menu, then choose the FlashCut CNC program group, then choose the FlashCut CNC program icon.

Using the Mouse

Most mice have two buttons. When using FlashCut CNC, always use the left mouse button unless specifically instructed otherwise. The following table explains basic terms associated with using the mouse.

To	Do this
Point	Position the pointer (arrow) on an item.
Click	Point to an item, and then quickly press and release the mouse button.
Double-click	Point to an item and then quickly press and release the mouse button twice.
Drag	Point to an item, press and hold the mouse button as you move the mouse to a new location. Then release the mouse button.
Hold	Point to an item, press and hold the mouse button.

Choosing Commands

A command is an instruction that tells FlashCut to perform a task. You can choose a command by:

1. Choosing a command from a menu with your mouse.
2. Choosing a command from a menu with your keyboard.
3. Using shortcut keys.
4. Using the TAB and ARROW keys.

Choosing a Command by Using the Mouse

Click the name of a menu item on the menu bar, then click the command name. To close the menu without choosing a command, click outside the menu.

Choosing a Command by Using the Keyboard

Press ALT or F10 to make the menu bar active, and then press the key corresponding to underlined letter in the menu name. To choose a command, press the key corresponding to the underlined letter in the command name. To close a menu without choosing a command, press ESC.

Using Shortcut Keys

Some of the FlashCut commands have shortcut keys associated with them. You can choose these commands by pressing the shortcut keys listed on the menu to the right of the command.

Using the TAB and ARROW Keys

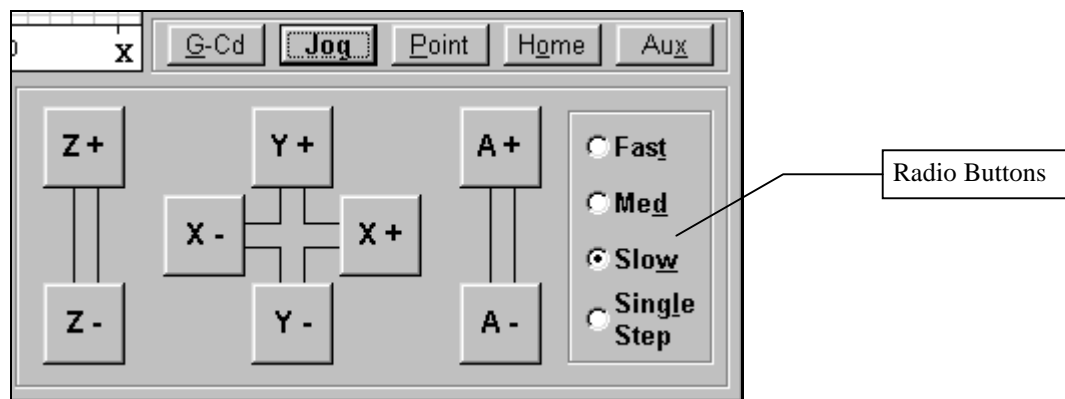
The TAB key and ARROW keys can be used to navigate through the currently active screen selections. Once the selection that you want is highlighted, use the RETURN key to select it.

Using Standard Windows Controls

FlashCut uses several standard Windows controls: **radio buttons**, **pull-down menus**, **text boxes** and **command buttons**.

Radio Buttons

Radio buttons represent a group of options, of which only one can be selected at a time.



Pull-Down Menus

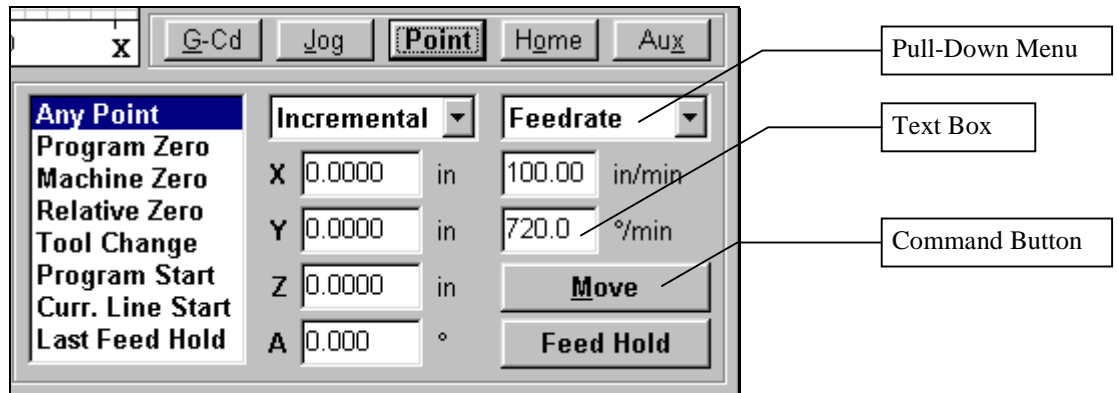
A pull-down menu is a list of commands that appears when you select either a menu or a down-arrow icon.

Text Boxes

Text boxes are areas in which you type either a name or a value.

Command Buttons

Command buttons perform a specific task when selected.



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:

1. Never let the machine tool run unattended.
2. 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.
3. 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.

4. 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.
5. Never place any part of your body within the tool envelope while the machine is online, since unexpected machine movement can occur at any time.
6. Always keep the tool envelope tidy and free of any loose objects.
7. Be on alert for computer crashes at all times.

WPI, Inc. is 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. WPI supplies this product but has no control over how it is installed or used. Always be careful!

WPI, Inc. 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.

About this Manual

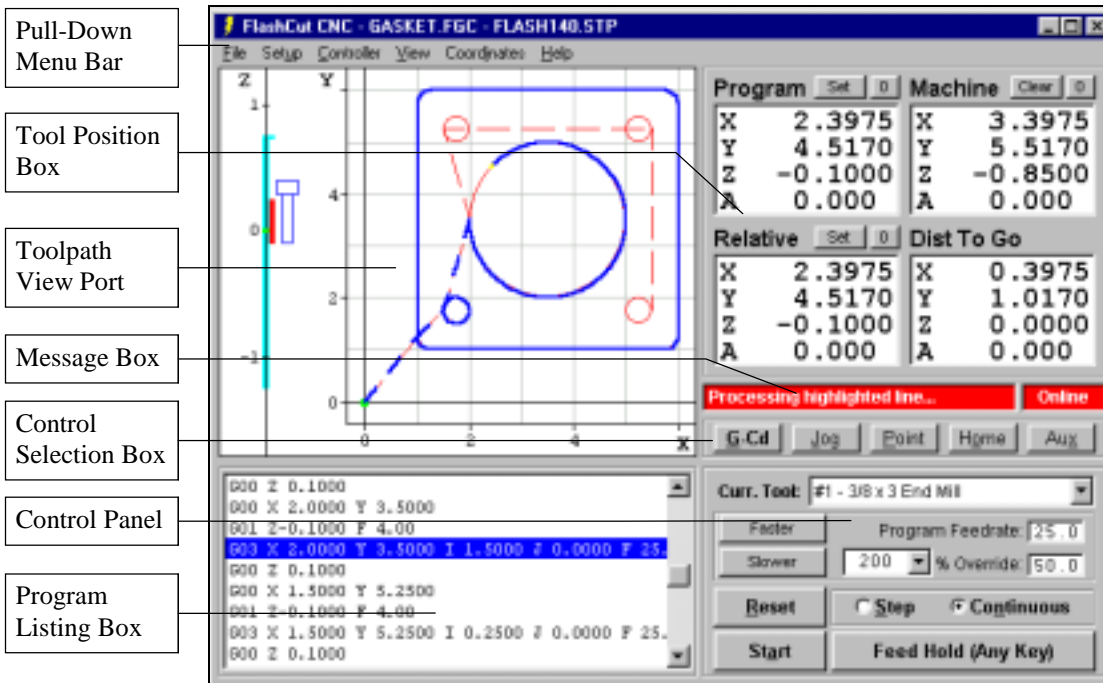
FlashCut CNC is a unique Windows application, so you'll need some instruction to get started. **Since automated machining is potentially dangerous, please take the time to completely read through this manual to understand operation of the software before running the system.**

Please note that all FlashCut CNC terminology appears in boldface upon first occurrence and is defined in the glossary.

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.

Section 2. Main Screen Features

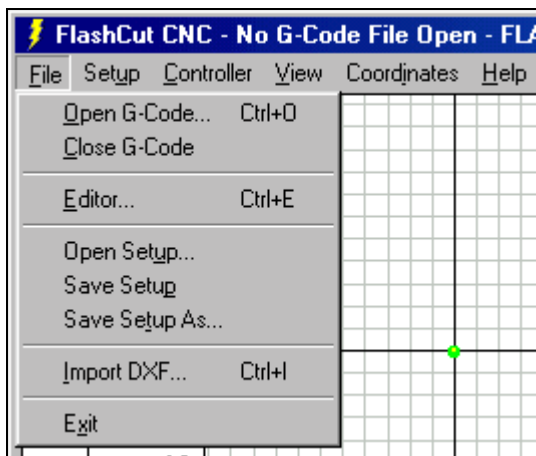
The main screen is shown below. An explanation of each area of the screen follows.



Pull-Down Menu Bar

This area contains the main menu headings for many system commands.

File Menu



Open G-Code - Opens an existing G-Code file, checks the file for errors and compatibility with FlashCut, draws the **toolpath** in the Toolpath View Port, and displays the program in the Program Listing Box. By default, the dialog box displays files with an ".FGC" extension.

Close G-Code - Closes the open G-Code file.

Editor - Opens the editor dialog box and displays the current G-Code file. Using this feature you can directly edit any G-Code file without leaving FlashCut. Note that you can also double-click the Program Listing Box to open the editor.

Open Setup - Opens a file containing all **CNC setup parameters** for your machine tool. You can set these parameters using commands in the Setup and View menus described below. **Setup file** names have a “.STP” extension by default.

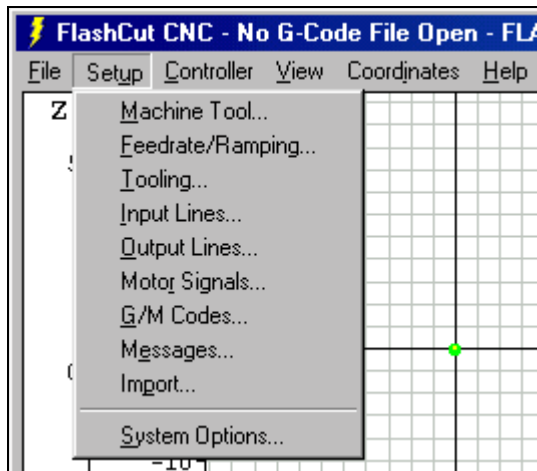
Save Setup - Saves the current CNC setup parameters under any file name in any directory.

Save Setup As - Prompts you for a file name in which to save the current CNC setup parameters.

Import DXF - Directly translates a 2-dimensional **DXF** file into G-Code for use in FlashCut.

Exit - Closes the communications port to the Signal Generator, then exits the program. **This does not automatically turn off the Signal Generator, motor driver or other electronics in your system. You must turn off all electronics separately.**

Setup Menu



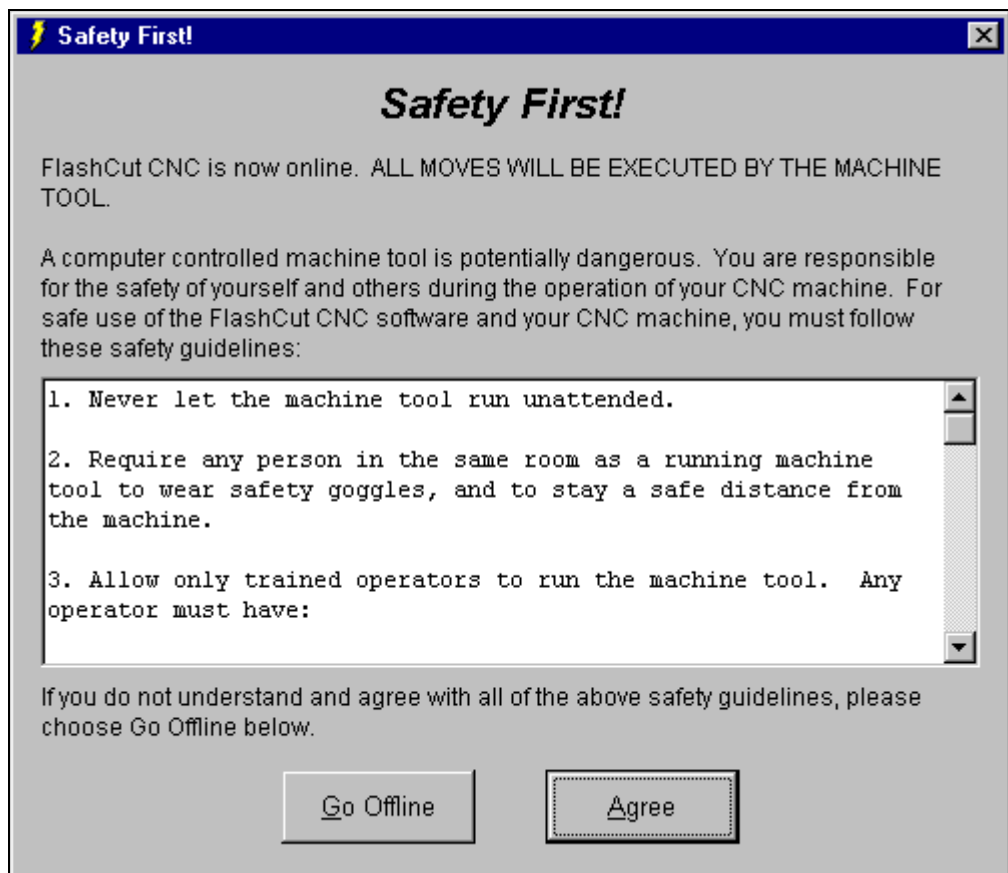
Machine Tool, Feedrate/Ramping, Tooling, Input Lines, Output Lines, Motor Signals, G/M Codes, Messages, Import, System Options - Open dialog boxes where you'll enter your CNC setup parameters, such as the size of your machine tool envelope or the configuration of your Signal Generator. These parameters are described in detail later in the manual.

Controller Menu



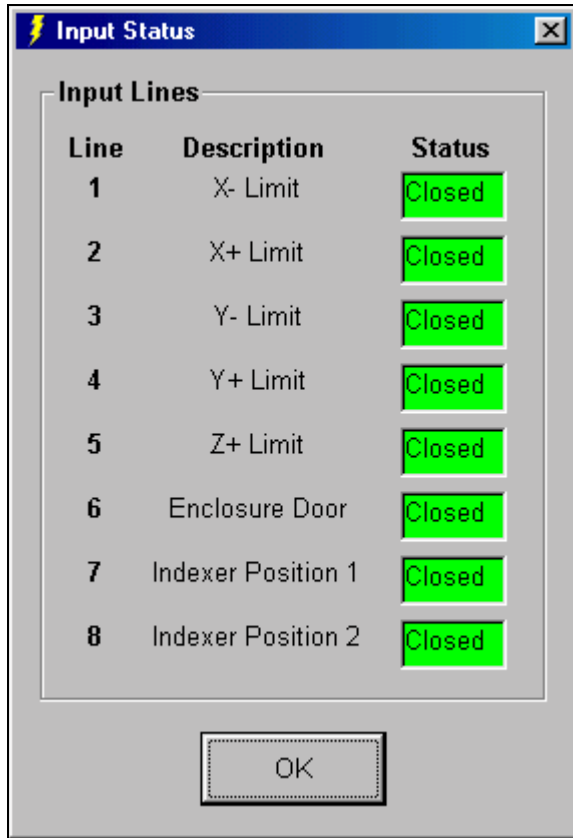
Online - Establishes communications with the Signal Generator. Once communications are established, FlashCut places a check mark next to this menu item. When the Signal Generator is online, all move commands will be executed by the machine tool, and the screen will update in real time.

Once the unit goes online, a safety reminder screen appears. It is imperative that you and anyone else near the machine understand, agree and adhere to all of the safety guidelines. If the safety guidelines are not agreed to, the unit will immediately go offline.

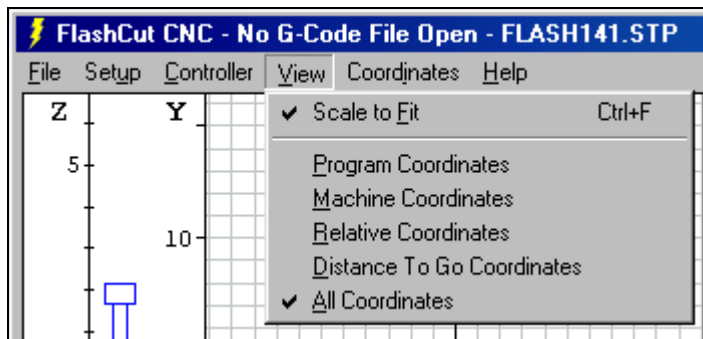


Offline - Breaks communication with the Signal Generator. When the Signal Generator is offline, FlashCut places a check mark next to this menu item. In this mode, the screen will update, but the machine tool will not move. This option lets you “animate” a G-Code file, which is useful for debugging before cutting a part.

Input Status - Shows the current status of the input lines. The following dialog box is displayed:

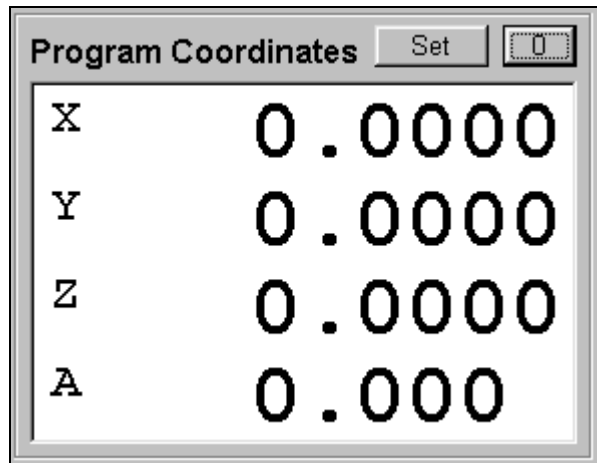


View Menu



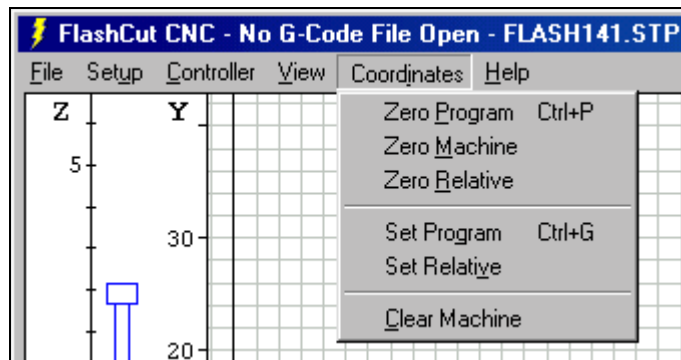
Scale to Fit - Causes the toolpath of the current G-Code File to expand as much as possible within the Toolpath View Port. When this option is not chosen, the Toolpath View Port displays the entire work envelope.

Program, Machine, Relative, Distance To Go, or All Coordinates - Selects the display mode for the Tool Position Box. Choosing either **Program Coordinates**, **Machine Coordinates**, **Relative Coordinates**, or **Distance To Go Coordinates** will expand the chosen Coordinate System Display into the entire Tool Position Box as shown below.



Choosing All Coordinates will display all four coordinate systems simultaneously in the Tool Position Box. You can also change these view modes by double-clicking any of the Coordinate System Displays.

Coordinates Menu



The commands in the Coordinates menu exactly match the buttons in the Tool Position Box. The main purpose of this menu is to provide keyboard equivalents and shortcuts for the buttons. See “Tool Position Box” later in this section for a complete explanation of each command.

Zero Program - Same as the Zero (0) button next to the Program coordinates label.

Zero Machine - Same as the Zero (0) button next to the Machine coordinates label.

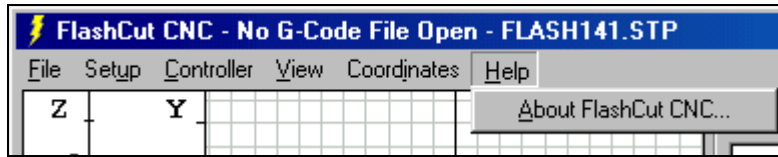
Zero Relative - Same as the Zero (0) button next to the Relative coordinates label.

Set Program - Same as the Set button next to the Program coordinates label.

Set Relative - Same as the Set button next to the Relative coordinates label.

Clear Machine - Same as the Clear button next to the Machine coordinates label.

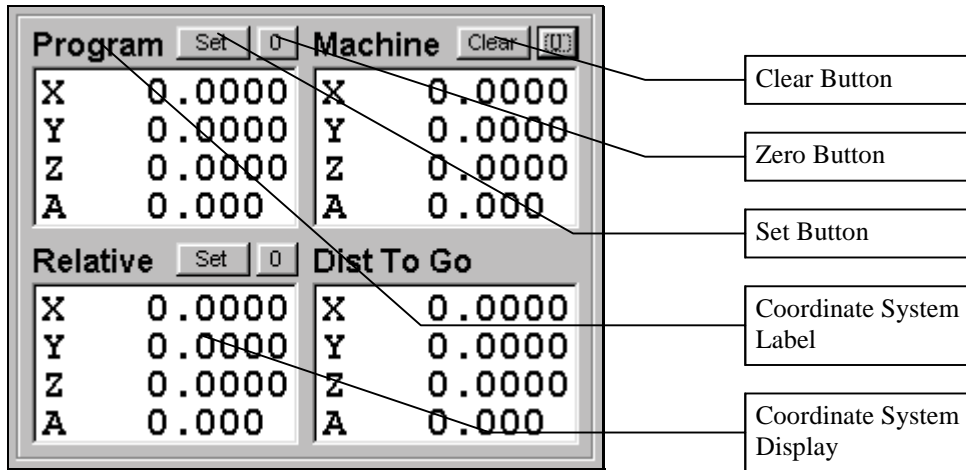
Help Menu



About FlashCut CNC - Shows the FlashCut CNC software version number.

Tool Position Box

The tool position box shows the current tool position in terms of Program, Machine, Relative and Distance to Go coordinates. It also has command buttons that affect the coordinate values.



Coordinate Systems

Program

Displays the coordinates of the current position of the tool relative to **Program Zero**.

Machine

Displays the coordinates of the current position of the tool relative to **Machine Zero**. This coordinate system is undefined if Machine Zero has not been set (displays "N/A").

Relative

Displays the current **relative coordinates**. The relative coordinate system is general purpose and may be used for anything you choose. For instance, to measure the distance from any point in the program without having to use the Program or Machine coordinates, just zero the relative coordinates at the point from which you want to measure.

Distance To Go

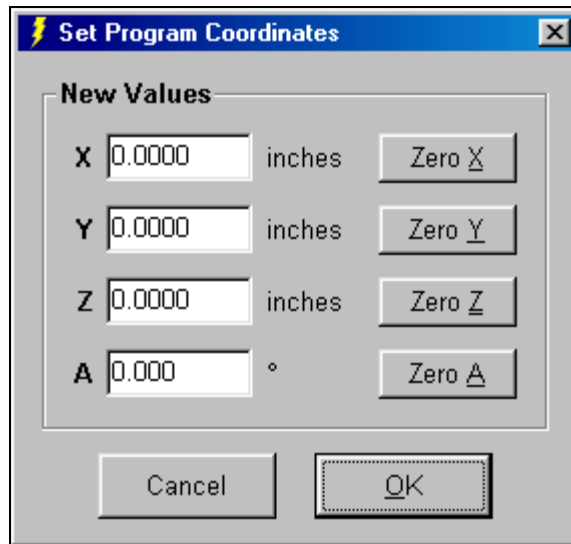
Displays the distance to the ending position of the current move.

Note that you can double-click any Coordinate System Display to expand it to fill the entire Tool Position Box.

Command Buttons

Zero Button - Sets X, Y, Z and A values of the indicated coordinate system to zero.

Set Button - Allows setting the X,Y Z and A values of either Program or Relative coordinates to any value. When chosen, the following dialog box appears:



▶ To set new values within a coordinate system

1. Type in the X, Y, Z and A values for each axis. These coordinates will become the current position of the tool.
2. Choose OK.

▶ To zero specific axes individually

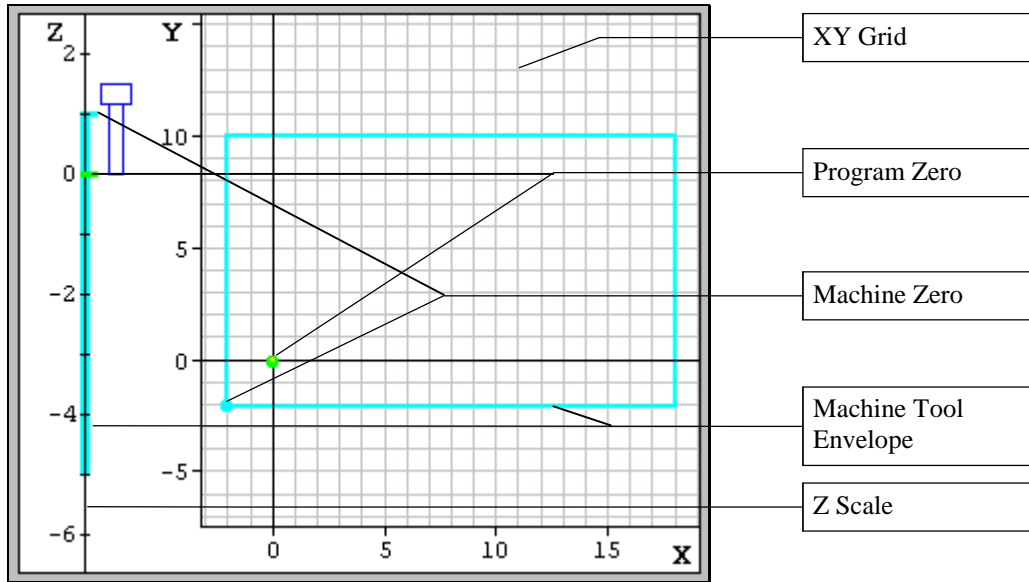
1. Choose the Zero button for each axis you want to zero.
2. Choose OK.

Clear Button - Clears the current Machine Zero setting. This button is useful when you've set Machine Zero manually (using the Machine coordinates Zero Button) and need to make a correction to the Machine Zero location.

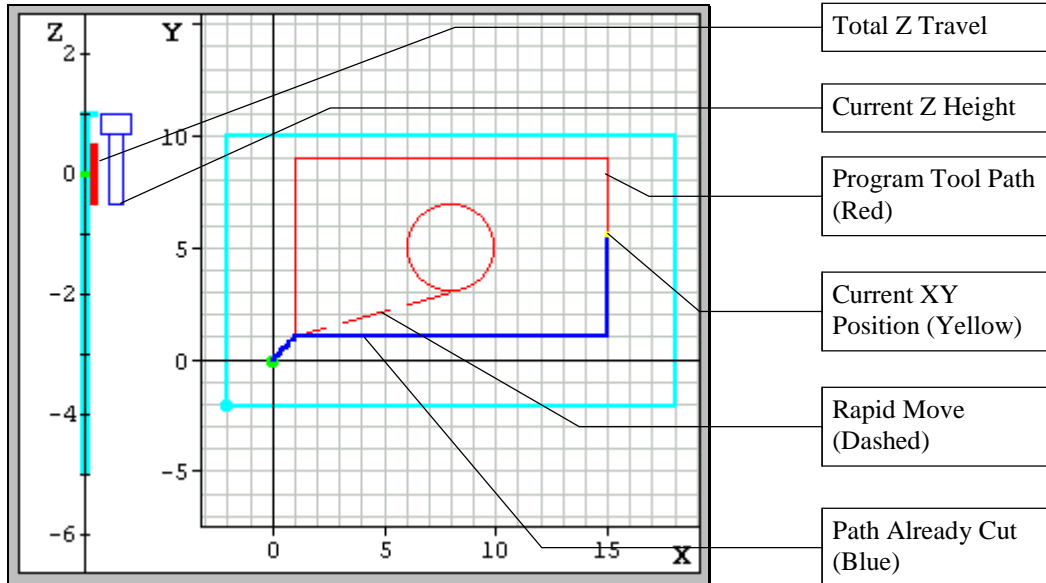
Toolpath View Port

The figure below shows the Toolpath View Port. The XY Grid represents an aerial view of the tool envelope. The Z Scale represents the height of the tool during machining. Green and light blue dots represent the origins of the Program and Machine (if used) coordinate systems respectively. The Machine Tool

Envelope is shown as the light blue box on the XY Grid and by the light blue bar on the Z Scale.

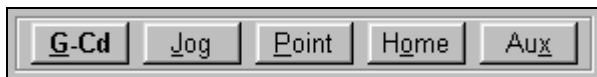


The figure below shows the Toolpath View Port during the machining process. The bottom of the blue tool icon adjacent to the Z scale represents the current Z position of the tool. The yellow dot on the XY Grid represents the current position of the tool. The red outline in the XY Grid represents the toolpath of the entire G-code program. The red bar along the Z Scale represents the total Z travel. Blue represents the portion of the toolpath that has already been cut. Solid lines depict **feedrate** moves while dashed lines represent rapid moves.



Control Selection Box

The Control Selection Box has buttons to display the five Control Panels.



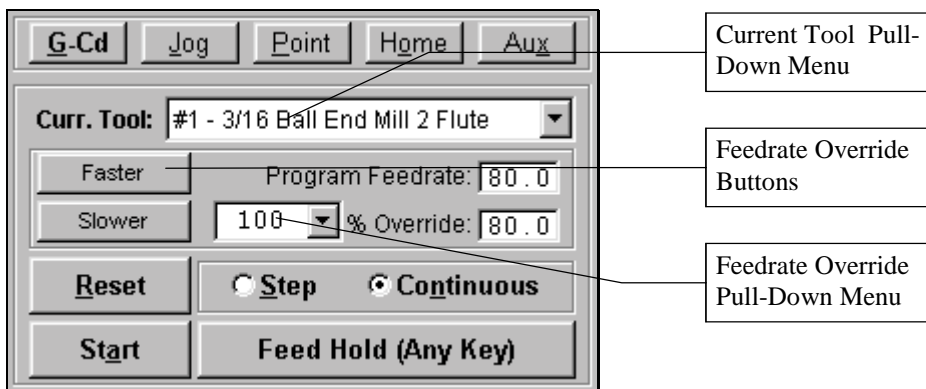
All of the controls you'll need to run your machine tool are located on these panels as follows:

- G-Cd Controls to move the tool along the toolpath specified by a G-Code program.
- Jog Controls to manually move the tool in all four axes.
- Point Controls to move the tool to any point specified.
- Home Controls to seek the home switches for all four axes.
- Aux Controls for turning auxiliary devices on or off.

To switch between panels, simply click on the appropriate button. Each control panel is described below.

G-Code Control Panel

The G-Code Control Panel provides controls to move the tool as directed by the current G-Code program.



Continuous / Step Radio Buttons - Sets the G-Code file processing mode. In continuous mode, the G-Code program runs non-stop. In Step mode, the G-Code program executes one line at a time. You can switch between step and continuous modes while the machine is moving. When **continuous contouring** is turned on, both buttons are disabled and the system always runs in continuous mode.

Start Button - Begins execution of the current line of the G-Code file. When in Step mode, execution will stop automatically at the end of the current line, or when the Feed Hold button is hit. When in Continuous mode, execution continues until the end of the program, or until the Feed Hold button is hit. If the program has been stopped in the middle of a G-Code line, choosing the Start

button will begin execution exactly where the program stopped. Note that all moves begin with ramping when necessary.

Feed Hold Button - Stops execution of the G-Code file. The machine tool will stop, ramping down if necessary to avoid losing steps. The slower the ramping rate, the longer it will take from the time the Feed Hold button is hit to the time the tool comes to a complete stop.

You can also stop the machine tool by hitting any key on the keyboard (except Shift or Ctrl).

Reset Button - Resets the current G-Code file to the first executable line and refreshes the Toolpath View Port.

Current Tool Pull-Down Menu - Displays the current tool loaded in the machine tool. Note that FlashCut uses this setting for tool length compensation (see System Programming for more information).

Feedrate Override Buttons and Pull-Down Menu - Increases or decreases the feedrate on the machine as a percent of the programmed feedrate. During linear interpolation moves (G01) you can gradually change the feedrate using the Faster or Slower buttons. The Faster and Slower buttons are disabled during circular interpolation and rapid moves. When the machine is stopped, you can enter an exact feedrate override percentage by choosing a value from the Feedrate Override pull-down menu (or typing in any value). Both the programmed feedrate and the override feedrate are displayed. Rapid moves are not affected by the feedrate override.

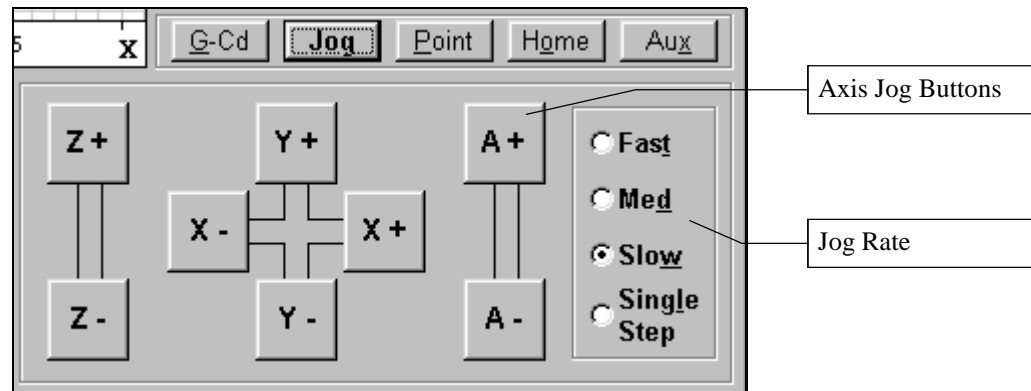
You can use the keyboard instead of the Faster/Slower buttons as follows:

Faster	Ctrl + Up Arrow Key
Slower	Ctrl + Down Arrow Key

When continuous contouring is turned on, the Faster and Slower buttons are disabled, and the Feedrate Override pull-down menu is disabled while a G-Code file is in progress. However, the Feedrate Override pull-down menu is available before running a G-Code file.

Jog Control Panel

The Jog Control Panel provides controls for manually positioning all axes.



Axis Jog Buttons

You can move a single axis of your machine tool by pressing and holding an Axis Jog Button. Ramping is used if the jog rate is faster than the maximum unramped feedrate for a given axis. Note that you can also jog the machine using the keyboard. The controls are mapped as follows:

X+	Ctrl + Right Arrow Key
X-	Ctrl + Left Arrow Key
Y+	Ctrl + Up Arrow Key
Y-	Ctrl + Down Arrow Key
Z+	Ctrl + Page Up Key
Z-	Ctrl + Page Down Key
A+	Ctrl + Plus (+) Key
A-	Ctrl + Minus (-) Key

Jog Rate

Single Step - Sets FlashCut to move exactly one **motor step** each time an Axis Jog button is chosen.

Slow, Medium and Fast - Sets the jog rate to the corresponding rate specified in the Feedrate/Ramping Setup dialog box. Note that there are separate jog rates defined for linear and rotary axis types.

Point Control Panel

The Point Control Panel provides controls for moving the tool to the XYZA position you enter at the feedrate you specify.

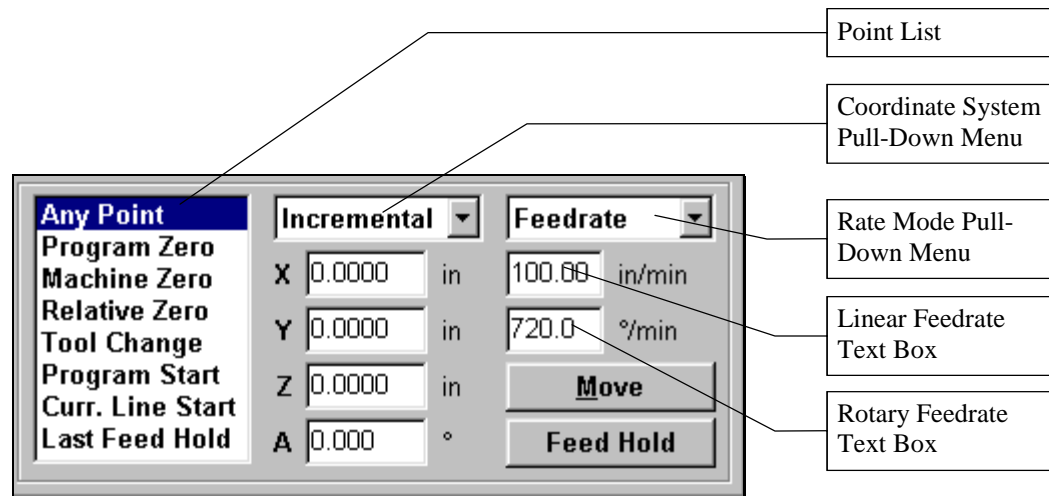
The sequence of axis motions follows a general-purpose scheme based on three fields in the Machine Tool Setup dialog box: Home End, Home Order, and Point Move Linear Interpolate. These fields should be configured to allow for safe moves in Point mode, which generally means the tool retracts from the workpiece before any other motion occurs. FlashCut uses the following sequencing rules:

1. Do all non-interpolated moves that are towards the Home End, in the Home Order.
2. Do the interpolated move, if any.
3. Do all non-interpolated moves that are away from the Home End, in reverse Home Order.

By setting the three Machine Tool Setup fields you can make the sequence safe for your machine tool configuration.

In a typical sequence for a 3-axis mill, any move with a positive Z axis element will first move up to the desired Z coordinate and then move across to the desired XY position. Any move with a negative Z axis element will first move across to the desired XY position and then move down to the desired Z position. This is the default behavior in the mill setup files provided with the FlashCut CNC software.

The G28 and G29 commands (described later in the System Programming section) follow the same scheme.



Point List

Provides a list of move options as follows:

Any Point - Moves to any XYZA point in the selected coordinate system.

Program Zero - Moves to Program Zero.

Machine Zero - Moves to Machine Zero (if defined).

Relative Zero - Moves to Relative Zero.

Tool Change Position - Moves to the Tool Change Position defined in Machine Coordinates in the Machine Tool Setup dialog box.

Program Start Point - Moves to where the tool was located when the current G-Code program was started.

Current Line Start Point - Moves to where the tool was located when the current G-Code line began execution.

Last Feed Hold Point - Moves to where the tool was located when G-Code execution was stopped by the Feed Hold button.

Note that you can select each item in the Point list using the keyboard as follows:

Any Point	Shift + 1
Program Zero	Shift + 2
Machine Zero	Shift + 3
Relative Zero	Shift + 4
Tool Change Position	Shift + 5
Program Start Point	Shift + 6
Current Line Start Point	Shift + 7
Last Feed Hold Point	Shift + 8

Coordinate System Pull-Down Menu - The tool will move to the XYZA position in program coordinates, machine coordinates, relative coordinates, or incrementally from the current position of the tool, depending on the option you select.

Rate Mode Pull-Down Menu - You can set the travel rate by selecting one of the following:

Rapid - The machine tool moves at the maximum feedrate allowed by your current **maximum feedrate** settings in the Feedrate/Ramping Setup dialog box.

Feedrate - The machine tool moves at the feedrate you enter in one of the feedrate text boxes.

Linear Feedrate Text Box - If Feedrate is selected in the Rate Mode pull-down menu, any moves that involve linear axes only will be done at this feedrate.

Rotary Feedrate Text Box - If Feedrate is selected in the Rate Mode pull-down menu, any moves that involve the A axis will be done at this feedrate.

Move Button - Executes the move.

Feed Hold Button - Stops execution of the move. Hitting any key on the keyboard (except Shift or Ctrl) also stops the move.

Home Control Panel

The Home Control Panel provides controls to seek the limit switches and establish the Machine Zero location.



Seek Home - Finds the home switch on up to four axes. To save time, it is recommended that you first jog each axis near the home switch before homing. Once Machine Zero (home) is set, the **machine tool envelope** is redefined.

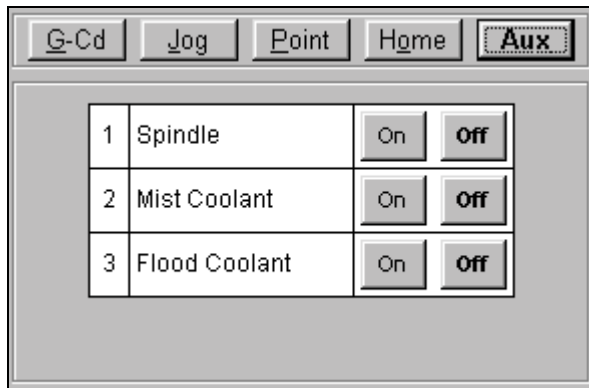
If Machine Zero was already set before homing, FlashCut displays a dialog showing the discrepancy between the previous Machine Zero and the new Machine Zero just found. This provides a convenient way to check that no steps were lost while cutting a part, within the accuracy limits of the home switch. The home switches supplied as a FlashCut accessory have a repeatability of +/-0.001”.

The homing operation can be tailored several ways, to suit the particular machine tool configuration (see “Machine Tool Settings” in the Initial Setup section of this manual.)

Feed Hold Button - Stops the homing operation. Hitting any key on the keyboard (except Shift or Ctrl) also stops the operation.

Auxiliary Control Panel

The Auxiliary Control Panel provides controls for turning on or off any auxiliary devices, such a spindle or coolant pump. Controls are displayed for each output line with a Description entered in the Output Lines Setup dialog box.



On / Off Buttons - Turns the indicated output line on or off. For safety, FlashCut displays a confirmation dialog when you turn on a device (click OK to proceed).

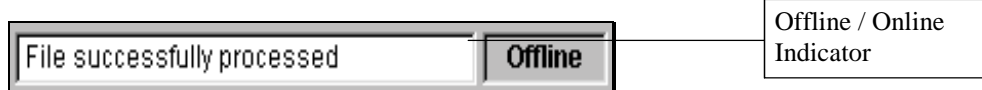
Note that you can operate the On / Off buttons using the keyboard as follows:

Ctrl + output line number turns ON the line

Alt + output line number turns OFF the line

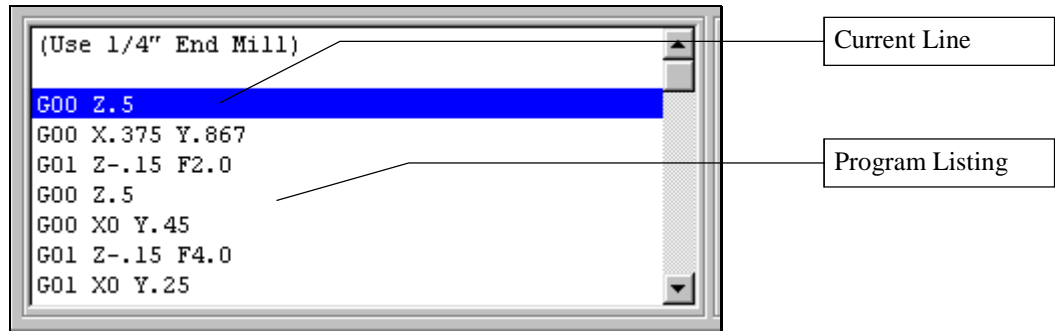
Message Box

Displays the current status of the Signal Generator and program. **When the Signal Generator is online, the Message Box and the Offline/Online indicator are Red.**



Program Listing Box

The Program Listing box displays the current **G-Code program** and highlights the current line. You can use the scroll bar to view the entire program.



Program Listing - A listing of the current G-Code program.

Current Line - The line currently being executed, or about to be executed by FlashCut.

To improve system performance, the Program Listing Box can be configured to display the current line of G-Code only, or to disappear completely while a G-Code file is running (see “System Settings” in the Initial Setup section of this manual).

Note that you can open the Editor dialog box by double-clicking the Program Listing Box.

Section 3. Initial Setup

This section describes how to set up FlashCut for use with your machine tool. It's very important that the software and hardware are set up properly. Incorrect setup 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.

Windows Setup

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.

▶ **To Disable the Screen Saver**

Windows 95, 98 or NT

1. Choose Settings, then Control Panel from the Start Menu.
2. Double-click the Display icon.
3. Select the Screen Saver tab.
4. Select “(None)” from the Screen Saver pull-down menu.
5. Choose the Apply button.
6. Choose OK to exit.

Windows 3.1

1. Double-click the Control Panel icon in the Main program group.
2. Double-click the Desktop icon.
3. Select “(None)” from the Screen Saver Name pull-down menu.
4. Choose OK to exit.

Software Setup

The Setup File

All software settings are stored in a “setup” file, which by default has a “.STP” extension. Before you start, you'll need to open the appropriate setup file for your

machine tool. Choose Open Setup from the File menu and select the appropriate setup file. Some setup files are supplied for various mills and lathes. If a setup file is not available for your machine, select FLASHXXX.STP, where “XXX” is the current software version (e.g. “FLASH141.STP” for version 1.41). You can configure the setup parameters in this file for your machine tool.

Once you’ve opened a setup file, choose Save Setup As and follow the dialog boxes to save the setup file under a new name or directory. This lets you go back to the original setup file if necessary. Using this method you can create unique setup files for any number of machine tools.

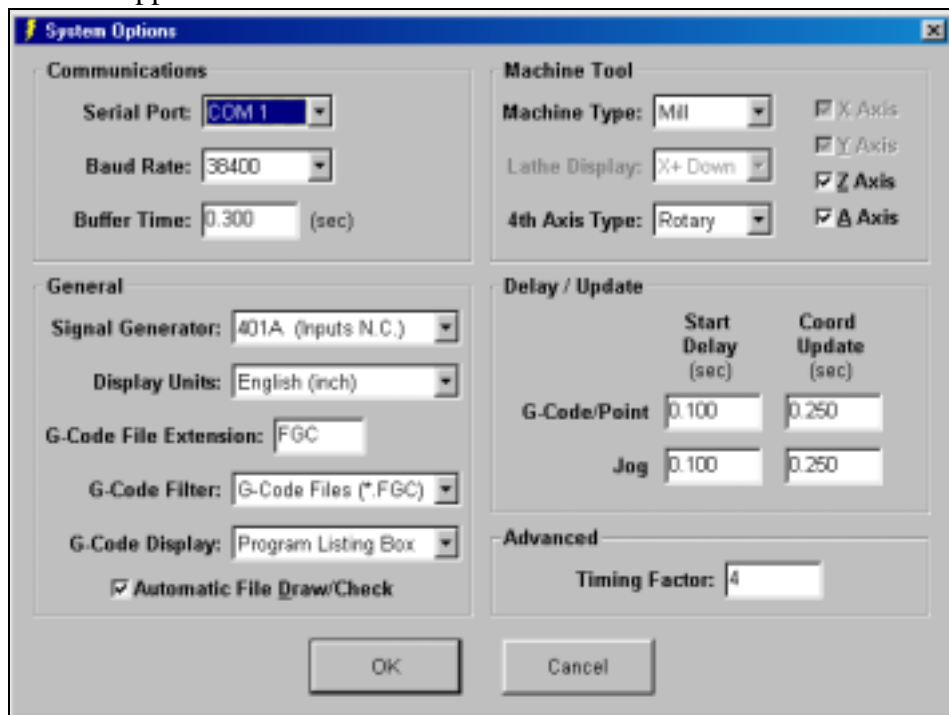
For machines that operate in more than one mode, such as combination mill-lathe machines, you’ll load a separate setup file for each mode of operation.

The last setup file used will be loaded automatically the next time you run FlashCut.

System Settings

► To set the System Settings

1. Choose System Options from the Setup menu. The System Options dialog box will appear.



2. **Serial Port** - The serial port your computer will use to communicate with the Signal Generator. Typically, this can be COM1, COM 2, COM 3, or COM 4 depending on how many serial ports and serial devices you have, such as a modem.
3. **Baud Rate** - The speed at which FlashCut communicates across the serial port with the Signal Generator. It is measured in bits per second. This is typically set at 38,400. For older PC’s exhibiting serial communications

problems, set this to a lower speed. For most applications, FlashCut will perform the same at baud rates of 19,200, 38,400 and 57,600. If you change this setting, you must reset the baud rate on the Signal Generator by going Offline and Online from the Controller menu. If there are communication problems, you must turn the Signal Generator off and on before going back online.

4. **Buffer Time** - Used to prevent system events (such as screen updates) from affecting motor movement on the machine tool. The larger the Buffer Time, the less effect system events have on motor movement. The smaller the Buffer Time, 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.
5. **Signal Generator** - Indicates how the limit switches are wired. Choose 401A if the limit switches are wired normally closed. Choose 401 if the limit switches are be wired normally open. All limit switch kits supplied by FlashCut are wired normally closed.
6. **Display Units** - Specifies units (English or Metric) that all screens will display, and the units assumed by default for all G-Code files. If a G-Code file uses units different from this setting, the G20 or G21 command must be included at the top of the G-Code file (see the System Programming section for details).
7. **G-Code File Extension** - Sets the file extension used when opening G-Code files. If you normally open files made by the FlashCut DXF import or files you've written in the FlashCut editor, set this to "FGC". If you normally open files made by another CAM program, enter its standard G-Code file extension (such as "NC").
8. **G-Code Filter** - Sets the filter used when opening G-Code files. Select "All Files" if you normally open files with a variety of extensions. Select "G-Code Files" if you normally use files with the G-Code File Extension described above.
9. **G-Code Display** - Affects how the software displays the Program Listing Box when running a G-Code file in continuous mode. When running G-code files with many short moves, or when continuous contouring is turned on, the FlashCut software runs faster if you select "Current G-Code Line," and runs fastest if you select "None".
10. **Automatic File Draw/Check** - Determines whether FlashCut redraws and rechecks a G-Code file each time you take action that affects processing of the file (such as resetting Program coordinates). If you uncheck this checkbox, FlashCut will not redraw and recheck the G-Code file automatically. You will need to reset the file manually just before running the file (using the Reset button on the G-Code Control Panel).

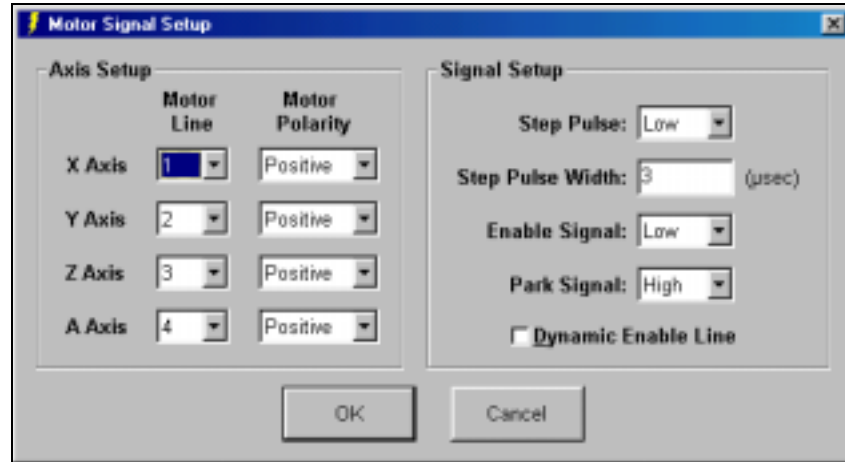
11. Machine Type - Configures the software for Mill or Lathe mode. Mill mode may be used for any XY or XYZ table including routers, plasma cutters, and so on.
12. X, Y, Z, A Axes - Sets the axes in use on your machine tool. Check the appropriate check boxes.
13. Lathe Display - Sets the orientation of the X+ direction in the Toolpath View Port. This setting lets you match the display in the Toolpath View Port to the actual direction of movement of the tool (for lathes only).
14. 4th Axis Type - Sets the 4th axis as either linear or rotary. In FlashCut, a rotary 4th axis is assigned letter 'A', while a linear 4th axis is assigned the letter 'W'. This manual will always refer to the 4th axis as the 'A' axis, but the 'W' alternative is always assumed.
15. Start Delay - Allows the button graphics to fully update after the Start, Move or Axis Jog Buttons are selected, and before the machine tool begins moving. There are separate values for Jog moves and G-Code or Point moves. Slower computers may require a higher value.
16. Coordinate Update - Determines how often the coordinates are updated in the Tool Position Box while the tool is moving. There are separate values for Jog moves and G-Code or Point moves. Higher values may increase performance on slower computers. Note that coordinates are always updated at the end of every move.
17. Timing Factor - Sets the precision for timing on interpolated moves. The higher the number, the greater the precision and the lower the system's top speed. For digital servo drivers, set this value to 2. For stepper drivers, set this value between 3 and 8.

Motor Signal Settings

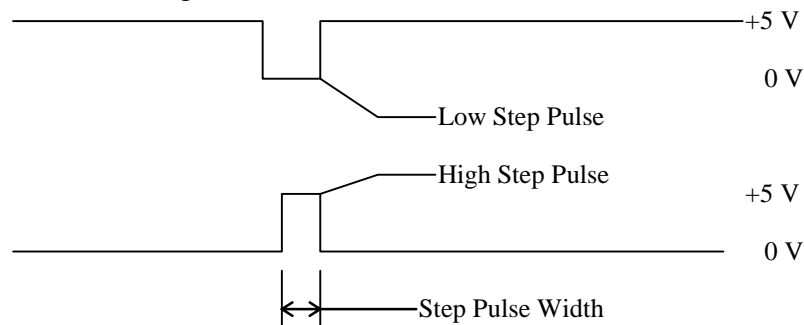
FlashCut provides four signals for motor drivers: step, direction, park and enable. Different manufacturers have different requirements for the polarity and timing of these signals. FlashCut provides the flexibility to tailor the motor signals to run most drivers.

► To set the Motor Signal Settings

1. Choose Motor Signals from the Setup menu. The Motor Signal Setup dialog box will appear.



2. **Motor Line** - Sets the Signal Generator channel used for the axis. This setting lets you map any axis letter to any channel in the Signal Generator. This is particularly useful when the axes on a machine tool require different axis letters depending on the mode of operation (e.g. combination mill-lathe machines).
3. **Motor Polarity** - Sets the direction of rotation for the motor. 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 Polarity from Positive to Negative (or vice-versa) to reverse the correspondence between the software and machine tool.
4. **Step Pulse** - Sets the polarity of the step signal. Choose High or Low based the diagram below.



5. **Step Pulse Width** - Sets the duration of the step pulse in microseconds.
6. **Enable Signal** - 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.
7. **Park Signal** - Sets the polarity of the park signal. Choose High if a high signal to the park (low power) line puts the driver into a reduced power mode. Choose Low if the opposite is true. Note that most motor drivers

do not have a separate line to control the power level. In this case the setting isn't applicable.

- 8. 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.

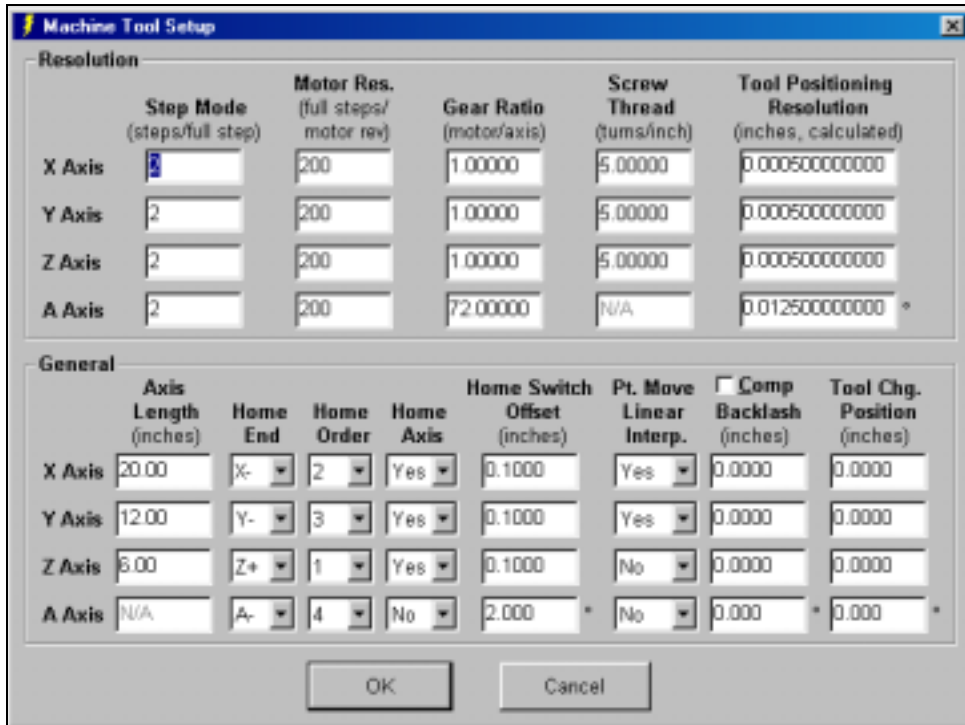
The recommended motor signal settings for FlashCut CNC Motor Driver Boxes are:

Motor Driver Type	Stepper	Servo
Step Pulse	Low	Low
Step Pulse Width	3	4
Enable Signal	Low	High
Park Signal	Not Used (Low or High)	Not Used (Low or High)
Dynamic Enable Line	Unchecked	Checked

Machine Tool Settings

► To set the Machine Tool Settings

- 1. Choose Machine Tool from the Setup menu. The Machine Tool Setup dialog box will appear.



1. **Step Mode** - The number of micro steps between each full motor step. Note that this is a characteristic of the motor driver and cannot be changed without servicing the driver. Enter “1” for **full-step**, “2” for **half-step**, “8” for eighth-step, and so on.
2. **Motor Resolution** - The number of full motor steps for one revolution of the motor. For example, a 1.8° **stepper motor** has 200 full steps per revolution, a 0.9° stepper motor has 400 full steps per revolution, and so on. This number is a characteristic of the stepper motor and is independent of the motor driver or the Step Mode.
3. **Gear Ratio** - The ratio of the number of stepper motor revolutions to drive screw revolutions due to any gears or pulleys between them. For direct drive, enter 1 in this box.
4. **Screw Thread** - The number of *turns* per unit length of the helical drive screw for each axis. For example, a single threaded, 0.05” pitch screw would have 20 turns per inch. For other types of drive mechanisms, enter the number of motor or pulley turns per unit length traveled by the axis.
5. **Tool Positioning Resolution** - The length of axis movement per motor step), automatically calculated from the Step Mode, Motor Resolution, Gear Ratio and Screw Thread.
6. **Axis Length** - Sets the length of travel 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.
7. **Home End** - The end of the axis at Machine Zero (Home).
8. **Home Order** - The axis order for the homing operation. For safety, set the order so the initial homing move retracts the cutting tool away from the workpiece.
9. **Home Axis** - Determines whether or not an axis is included in the homing operation.
10. **Home Switch Offset** - The distance each axis backs away from the home switch after the switch is tripped during homing.
11. **Point Move Linear Interpolate** - Determines if the motion for the axis will be included in the interpolated move for a Point move or G28/G29 command (see “Point Control Panel” in the Main Screen Features section of this manual).
12. **Backlash** - Sets the amount of backlash for each axis. See the “Setting Up Backlash” section below for more information.
13. **Comp** - Determines whether or not FlashCut will use backlash compensation for all direction changes. Leave this check box unchecked for now. It’s discussed in the “Setting Up Backlash” section below.

14. Tool Change Position - The position to which FlashCut moves the machine tool when processing a G28 command in a G-Code program. You must specify these values in Machine Coordinates. Machine Zero is the most common choice for this position.

Feedrate and Ramping Settings

Every machine tool will vary as to how fast it can move each axis without losing steps. Losing steps means that even though the stepper motor gets the signal to move a step, it isn't able to move the step, and accuracy is lost. 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 not "over torqued", a stepper motor is very reliable and accurate. For that reason, we highly recommend finding the maximum rates at which steps are not lost, both with and without ramping, and then limiting the maximum rates for each axis to about 70% of those values. Due to variations in the drive mechanism for each axis, make sure you do the following tests in all directions, and at several positions along each axis.

► To Set the Maximum Unramped Feedrates

1. Choose Feedrate/Ramping from the Setup menu. The Feedrate/Ramping Setup dialog box will appear.

Axis	Max Unramped Feedrate (in/min)	Maximum Feedrate (in/min)	Ramping Rate (full steps/sec/sec)	Direction Change Delay (sec)	Continuous Contouring Feedrate Tol. (in/min)
X Axis	15.00	150.00	5000	0.100	15.00
Y Axis	15.00	150.00	5000	0.100	15.00
Z Axis	15.00	150.00	5000	0.100	15.00
A Axis	360.0 °/min	1440.0 °/min	5000	0.100	360.0 °/min

Jog Rate Setup	
Linear	Rotary
Slow: 2.00 (in/min)	360.0 (°/min)
Med: 50.00 (in/min)	720.0 (°/min)
Fast: 150.00 (in/min)	1440.0 (°/min)

General	
Max Arc Feedrate:	100.00 (in/min)
PL Feedrate - Linear:	100.00 (in/min)
PL Feedrate - Rotary:	720.0 (°/min)
<input type="checkbox"/> Continuous Contouring On	

2. Enter 499 for the X axis Max Unramped Feedrate, and 500 for the Maximum Feedrate, then choose OK.
3. Choose the Point button in the Control Selection Box. Select Any Point in the Point list and Incremental from the Coordinate System pull-down

menu. Enter 1.0 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, then choose Move.

4. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate. Note that slight slippage can be detected by reading the values on the table feed hand-wheel or using a dial indicator.
5. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.
6. Now run the entire length of the X axis in both directions to make sure there is no slippage at any point on the entire axis.
7. Choose Feedrate/Ramping from the Setup menu.
8. Enter **70%** of the value you found in the Max Unramped Feedrate text box for the X axis, then choose OK.
9. Repeat this process for all axes.

► **To Set the Maximum Feedrates**

After finding the maximum unramped feedrates, you're ready to find the maximum feedrates achievable with ramping.

1. Choose Feedrate/Ramping from the Setup menu. The Feedrate/Ramping Setup dialog box will appear.
2. Enter 5000 full steps/sec/sec in the Ramping Rate text box for the X axis. (This is an average ramping rate.)
3. Leave the Max Unramped Feedrates at the values you found earlier. Leave the Maximum Feedrates at the very high number (500 in/min) and choose OK.
4. 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 in the X text box. Choose Feedrate from the Rate Mode pull-down menu and enter a feedrate that is double the Max Unramped Feedrate for the X axis. Make sure you have room to move the X axis 1 inch, then choose Move.
5. 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 by reading the values on the table feed hand-wheel or using a dial indicator.
6. Repeat this process until you find the highest feedrate that doesn't cause motor slippage.
7. 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.

8. Choose Feedrate/Ramping from the Setup Menu.
9. Enter **70%** of the highest no-slip feedrate you found in the X axis Maximum Feedrate text box.
10. Repeat this process for all axes.

▶ **To Set the Ramping Rate**

Ramping Rates typically range from 1000 to 100000 full steps/sec/sec. 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.

1. Choose Feedrate/Ramping from the Setup menu. The Feedrate/Ramping Setup dialog box will appear.
2. Enter 5000 full steps/sec/sec in the Ramping Rate text box for the X axis. (This is an average ramping rate.)
3. Leave the Max Unramped Feedrates and Maximum Feedrates at the values you found earlier and choose OK.
4. 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 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, 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.
5. 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 by reading the values on the table feed hand-wheel or using a dial indicator.
6. Repeat the above steps until you determine an optimal Ramping Rate for the X axis.
7. Choose Feedrate/Ramping from the Setup menu.
8. Enter **70%** of the highest no-slip ramping rate you found in the X axis Ramping Rate text box.
9. Repeat this process for all axes.
10. 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.

▶ **To Set the Direction Change Delay**

The direction change delay is a brief pause that occurs when a motor changes direction. It gives the stepper motor time to settle and come to a complete rest before moving in the opposite direction. Note that the direction change delay is not used when a motor changes direction during circular interpolation or during a continuous contouring motion.

1. Write a G-Code file that goes back and forth on a given axis at the maximum unramped feedrate. For example, if the maximum unramped feedrate were 8:

```
G01 X2 F8
```

```
X0
```

```
X2
```

```
X0
```

2. Set the Direction Change Delay to 0.1 seconds for the given axis.
3. Run the program and notice if the motor loses steps. If so, increase the Direction Change Delay. Otherwise decrease the value.
4. Repeat the above process until you reach a reasonable delay time that eliminates any motor slippage. Note that this number is typically between 0.0 and 0.1 seconds. If you do not see any slippage at a delay of 0 seconds, it is recommended you enter at least 0.01 seconds.
5. Repeat the above process for all axes.

▶ **To Set the Continuous Contouring Feedrate Tolerance**

1. Choose Feedrate/Ramping from the Setup menu.
2. For each axis, enter the Maximum Unramped Feedrate value as a starting point. Refining this setting is explained later in this section.

▶ **To Set the Jog Rates**

1. Choose Feedrate/Ramping from the Setup menu.
2. Fill in the Slow, Medium and Fast Linear Jog Rates (and the corresponding Rotary Jog Rates if your machine tool includes a rotary axis). Choose a Slow Jog Rate that will allow fine positioning of the machine tool (typically about 2-5 in/min). Choose a Fast Jog Rate that will move the tool quickly, yet allow you to remain in complete control without creating a dangerous situation. Note that the machine tool will not jog any faster than the Maximum Feedrate for each axis.

▶ **To Set the Maximum Arc Feedrate**

Due to the computations involved during circular interpolation, an arc cannot be executed as fast as a line. For this reason, there is a user-configurable limitation for the maximum arc feedrate.

1. Write a G-Code program that moves in a circle at the Maximum Feedrate. For example, if the Maximum Feedrate were 100:

```
G00 X2 Y2
```

```
G02 X2 Y2 I1 J1 F100
```

2. Run the program and notice if either of the motors loses steps. If so, decrease the feedrate in the program.
3. Repeat the above process until neither motor loses any steps. Then enter the final feedrate from the program into the Maximum Arc Feedrate text box. If there was no loss of steps when the Maximum Feedrate was used, enter the Maximum Feedrate into the text box.

▶ **To Set the Point Feedrates - Linear and Rotary**

The value entered in the Pt. Feedrate - Linear and Pt. Feedrate - Rotary text boxes will appear in the Linear Feedrate and Rotary Feedrate text boxes on the Point Control Panel, whenever you start the FlashCut program. Enter the feedrates you most commonly use for Point moves. A good starting point is the medium jog rate.

▶ **To Set the Continuous Contouring On Check Box**

Continuous contouring (also called “Look-ahead”) is a high performance operating mode that lets FlashCut run G-code files smoother and faster. The advantages of using continuous contouring are:

1. 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.
2. 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 turned on.

In FlashCut, the differences between continuous contouring turned on and off are as follows:

Continuous Contouring Off

FlashCut handles all moves individually, independent of the prior and subsequent moves.

The feedrate override controls on the G-Code Control Panel are always available.

You can choose between Step and Continuous modes (radio buttons on the G-Code Control Panel).

Continuous Contouring On

FlashCut handles all moves as segments of smooth motions. A smooth motion may span hundreds or thousands of G-code lines (there is no limitation).

The feedrate override controls are not available while a G-Code file run is in progress.

Step mode is not available.

To determine if two moves are lined up enough to allow a smooth transition without ramping, FlashCut calculates the speed of each motor before and after the transition, and compares that to the setting for Continuous Contouring Feedrate Tolerance. If the difference in speeds 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 Maximum Unramped Feedrate 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.

Since Step mode and the feedrate override controls aren't available when continuous contouring is turned on, it's sometimes helpful to run a few lines of a new program with continuous contouring turned off, to allow adjustment of the feedrate override setting. Note that these controls will be made available in continuous contouring mode in a future upgrade to FlashCut.

Setting Up Backlash Compensation

CNC control works best if there is little or no backlash on the machine tool. However, if your machine doesn't have a zero-backlash drive mechanism (such as ball screws or anti-backlash nuts), FlashCut can compensate for the backlash.

► To Set Backlash

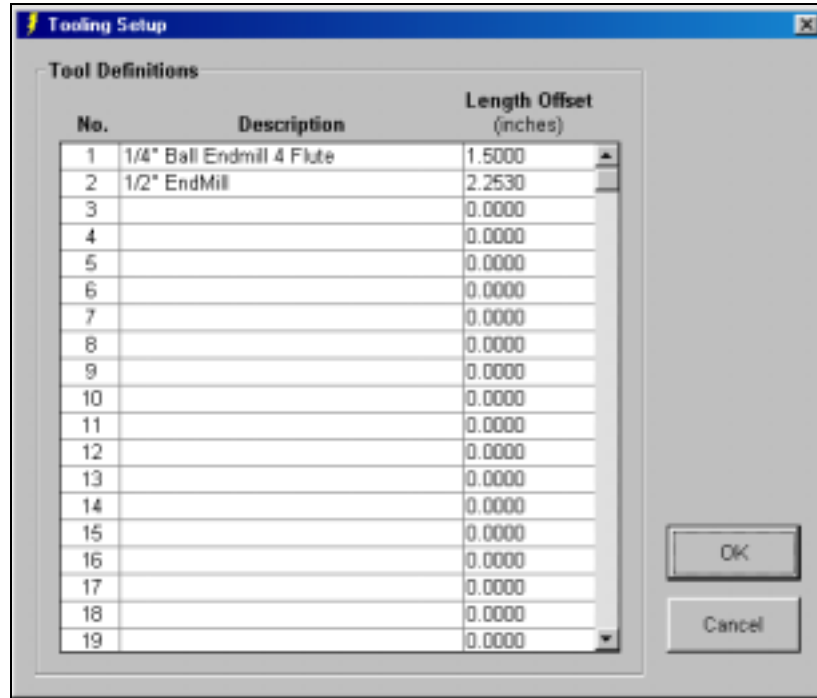
1. Choose Machine Tool from the Setup menu. The Machine Tool Setup dialog box will appear.
2. Make sure the Comp checkbox is unchecked, then choose OK.
3. Choose the Jog button on the Control Selection Box.
4. Drive the X axis in either direction at least 0.25" (to take out any backlash in that direction).
5. Zero the Relative coordinates by choosing the Zero button next to the Relative label.
6. Choose the Single Step radio button. Jog the axis step by step in the opposite direction until you detect table movement (using a dial indicator).
7. The Relative coordinate X axis value is the amount of X axis backlash on your machine tool.
8. Write down this number and repeat the above process at different places along the X axis.
9. Choose Machine Tool from the Setup menu. The Machine Tool Setup dialog box will appear.
10. Record the average of all backlash values in the X axis Backlash text box. If you have no backlash on an axis, or if you don't want backlash compensation on an axis, enter zero.
11. Repeat the above steps for each axis. When you're finished, select the Comp checkbox and make sure there is a check in it.

Tooling Settings

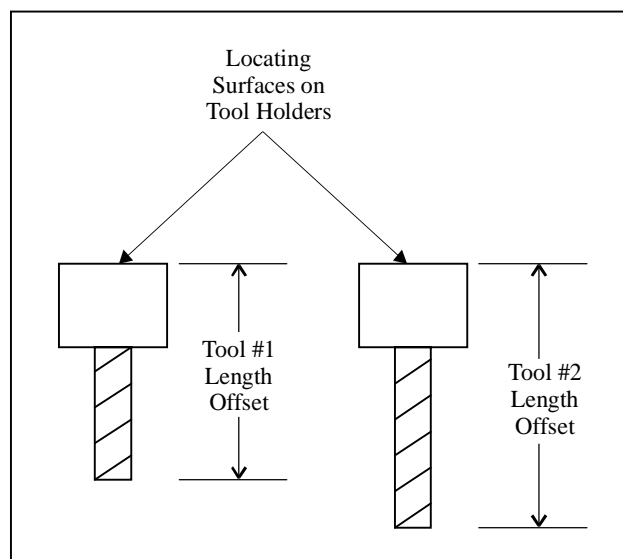
FlashCut provides for a tool library of up to 100 tools. Each tool has an associated tool number, description and length offset. The length offset is used when tool length compensation (G43 or G44) is used in a G-Code program.

► To Set Up the Tool Library

1. Choose Tooling from the Setup menu. The Tooling Setup dialog box will appear.



- Enter the tool description for each tool next to the appropriate tool number. The program will use this description when referring to a given tool.
- Enter the length of the tool relative to a common reference point. For example, if there is a locating feature on a tool holder that can be used to maintain a repeatable location between the tool holder and the spindle, enter the axial distance between the tip of the tool and the tool holder's locating feature. Note that this method assumes each tool has its own tool holder.



- Repeat the above for each tool in your library.

Input Line Settings

FlashCut can read up to 8 input lines wired to switches with various functions:

- Limit/home sensing
- General safety (such as a door switch on a safety enclosure or a servo error line)
- Control (such as a switch that detects completed motion of an air cylinder or other device).

You can use each of the 8 input lines however you choose, but all must be wired the same, either all normally open or all normally closed. If you're using the Model 401A Signal Generator, you may choose between normally open (N.O.) and normally closed (N.C.) wiring. If you're using the Model 401 Signal Generator, you must wire all input lines normally open. Make sure the Signal Generator setting in the System Options dialog box matches your wiring scheme.

Please see the FlashCut CNC Hardware Guide for more hardware details, including the pin layout of the input line connector.

When FlashCut detects a tripped limit or safety switch, it immediately stops all tool movement without ramping. The accuracy of the tool position will most likely be lost at that point.

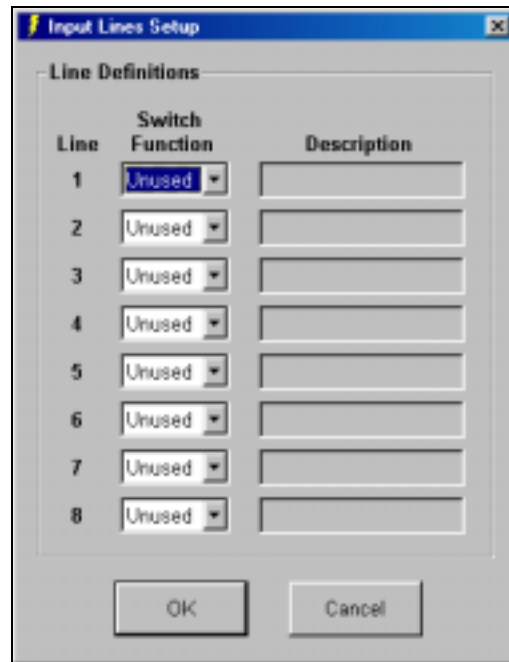
If a limit switch has been tripped, FlashCut will only allow you to jog away from the switch that was tripped. Once you move the table off the limit switch, normal operation will resume.

If a safety switch is tripped, FlashCut will not allow any machine movement until you clear the switch.

For information on how FlashCut handles control switches, please see “M100, M101 Wait for Input Line” in the System Programming section of this manual.

► To Configure the Input Lines

1. Choose Input Lines from the Setup menu. The Input Lines Setup dialog box will appear.



2. For input line 1, choose the appropriate option from the Switch Function pull-down menu. For example, if the line is wired to a limit switch on the negative end of the X axis, choose X-. If the line is wired to a general-purpose safety switch (including a servo error line), choose Safety and enter a Description. If the line is wired to a switch used for control purposes, choose Control and enter a Description (see “M100 / M101” in the System Programming section for more details on use of control switches). If the line is not used, choose Unused.
3. Repeat for all 8 input lines.

Output Line Settings

FlashCut can control up to 8 output lines to activate devices such as a spindle or coolant pump. You can control any or all of the output lines with user defined M codes. Please see the FlashCut CNC Hardware Guide for more hardware details including the pin layout of the output line connector.

► To Configure the Output Lines

1. Choose Output Lines from the Setup menu. The Output Lines Setup dialog box will appear.

Output Lines Setup

Output Lines

Line	Description	Line	Description
1	Spindle	5	
2	Mist Coolant	6	
3	Flood Coolant	7	
4		8	

M Codes

M Code	Description	Output Line Action								Before/After Move	Delay (sec)
		1	2	3	4	5	6	7	8		
03	Spindle On	On	-	-	-	-	-	-	-	Before	5.000
05	Spindle Off	Off	-	-	-	-	-	-	-	After	0.000
07	Mist Coolant On	-	On	-	-	-	-	-	-	Before	3.000
08	Flood Coolant On	-	-	On	-	-	-	-	-	Before	3.000
09	Coolant Off	-	Off	Off	-	-	-	-	-	After	0.000
06	Tool Change	Off	Off	Off	-	-	-	-	-	Before	0.000
		-	-	-	-	-	-	-	-	Before	0.000
		-	-	-	-	-	-	-	-	Before	0.000

- In the Description text boxes, type descriptions for each device connected to the output lines. Any line with a Description entered will appear on the Auxiliary Control Panel.

► **To define an M-code to control output lines**

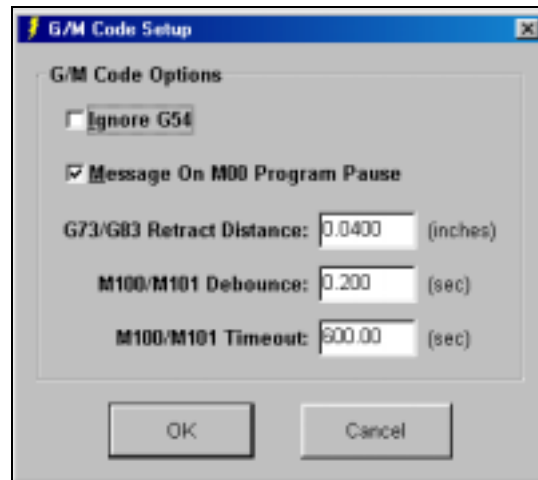
- In the M Code text box type the number for the M-code that you want to define.
- In the Description text box type a phrase that best describes the action taken when this M-code is executed.
- In the Output Line Action pull-down menus, choose the action for each line upon execution of the M-code. The choices are On, Off, or '- ', where '- ' indicates the state of the line remains unchanged.
- Choose Before or After from the Before/After Move pull-down menu. If you choose Before and there is a machine tool move command on the same program line as the M-Code, FlashCut will execute the M-code before the move. If you chose After, FlashCut will execute the M-code after the move.
- In the Delay text box, enter the amount of delay between execution of the M-Code and execution of the next G-Code command. For example, if the spindle motor takes about 3 seconds to reach full speed, this value should be at least 3. For safety reasons, the maximum value for this delay is 5 seconds.

G and M Code Settings

FlashCut lets you customize handling of some G and M codes.

► To Configure G and M Code Handling

1. Choose G/M Codes from the Setup menu. The G/M Code Setup dialog box will appear.



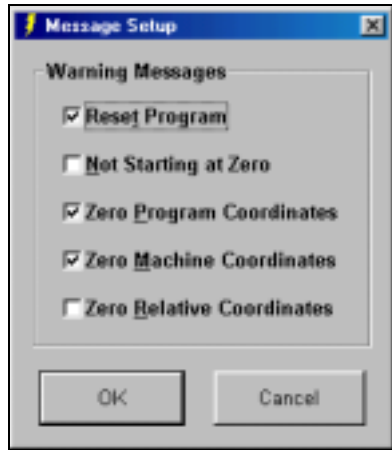
2. Ignore G54 - Directs FlashCut to ignore the G54 command in a G-Code program. If you select this option, make certain any G-Code program you run does not rely on G54 to position the machine tool. FlashCut does not currently support the G54 command.
3. Message on M00 Program Pause - Directs FlashCut to display a message dialog whenever it encounters an M00 command in a G-Code program. If you uncheck this option, FlashCut will still pause processing of the G-Code program on M00, but will not display a message to the operator.
4. G73/G83 Retract Distance - Sets the 'D' distance for canned drilling cycles. See "G73, G80, G81, G83, G98, G99 Canned Cycles" in the System Programming section for details.
5. M100/M101 Debounce and Timeout - Sets the debounce and timeout duration for the M100 and M101 commands. See "M100, M101 Wait for Input Line" in the System Programming section for details.

Message Settings

FlashCut lets you turn off some warning messages. This is a convenience feature intended for experienced FlashCut users only. Please do not turn off any warning messages until you're very familiar with operation of the software and your machine tool.

► **To Configure Warning Messages**

1. Choose Messages from the Setup menu. The Message Setup dialog box will appear.



2. Reset Program - Directs FlashCut to display a warning message whenever you choose Reset on the G-Code Control Panel.
3. Not Starting at Zero - Directs FlashCut to display a warning message whenever you choose Start on the G-Code Control Panel, and the Program coordinates have not been zeroed.
4. Zero Program Coordinates, Zero Machine Coordinates, Zero Relative Coordinates - Directs FlashCut to display a warning message whenever you choose the Zero button for the associated coordinate system.

Setting Machine Zero

FlashCut lets you set Machine Zero two ways: Using limit switches or using the Zero button. Either method establishes the machine tool envelope, which provides these benefits:

1. The software will always ramp down the machine tool to a complete stop at the defined limits of the machine tool envelope.
2. The software checks each G-Code file before running to make sure the toolpath fits completely within the machine tool envelope.

It's better to use limit switches since they provide these added benefits:

1. It's much more convenient, since FlashCut can find Machine Zero automatically.
2. You can establish Machine Zero at the same repeatable location anytime you like (within the repeatability tolerance of the switch). A repeatable Machine Zero lets you restart unfinished jobs, and reliably position the tool relative to work-holding fixtures.
3. You can use the homing process to check for accuracy of the machine tool's position.

▶ **To Set Machine Zero Using Limit Switches**

1. Make sure you've correctly entered the homing-related setup parameters described in the "Machine Tool Settings" and "Input Line Settings" sections above.
2. Choose the Home button on the Control Selection Box.
3. Choose the Seek Home button. One axis at a time, FlashCut drives the axis until it finds and backs off the switch. To stop the process, choose the Feed Hold button, or hit any key on the keyboard besides Ctrl or Shift.
4. If Machine Zero was already set before homing, FlashCut displays a dialog showing the discrepancy between the previous Machine Zero and the new Machine Zero just found. Choose Yes if you want to use the new Machine Zero just found, or No if you want to keep the existing Machine Zero.
5. If you chose Yes, FlashCut zeros the Machine Coordinates and displays the Machine Tool Envelope in the Toolpath View Port.

▶ **To Set Machine Zero Using the Zero Button**

1. Make sure you've correctly entered the homing-related setup parameters described in the "Machine Tool Settings" section above.
2. Choose the Jog button on the Control Selection Box.
3. Jog the tool to the home end of each axis (as defined in the Home End column in the Machine Tool Setup dialog box). Move each axis to within a short distance of its physical limit.
4. If this is the first time you're setting Machine Zero on your machine, it might be helpful to scratch a reference line or affix a vernier scale between the two relative moving parts on each axis. This will help you find the same home position again in the future.
5. Choose the Zero button next to the Machine label in the Tool Position Box. When the confirmation dialog appears, choose Yes (see "Message Settings" above for details on suppressing this dialog if desired).
6. If Machine Zero was already set, FlashCut displays a dialog showing the discrepancy between the previous Machine Zero and the new Machine Zero just set. Choose Yes if you want to use the new Machine Zero just set, or No if you want to keep the existing Machine Zero.
7. If you chose Yes, FlashCut zeros the Machine Coordinates and displays the Machine Tool Envelope in the Toolpath View Port.

Section 4. System Programming

FlashCut reads a subset of ANSI standard G-Code to control machine tool movement. This section describes how to bring a G-Code file into FlashCut, the G-Codes supported, and a brief explanation of their use.

There are three ways you can bring G-Code files into FlashCut:

- Open an existing G-Code file created by a CAM program, FlashCut or any other source.
- Import a DXF file created by a CAD or drawing program.
- Write a G-Code program directly in the FlashCut editor.

Opening a G-Code Program

► To open an existing program

1. Choose Open G-Code from the File menu. The Open G-Code File dialog box appears.
2. In the “List files of type” pull-down menu, choose the type of file you are looking for. Existing FlashCut files will have an “.FGC” extension. If you are unsure of the file type, choose “All Files (*.*)”.
3. In the Drives pull-down menu choose the drive that contains the file.
4. In the Folders list box, double-click the name of the folder that contains the file. Continue double-clicking subfolders until you open the subfolder that contains the file.
5. In the box that lists files, double-click the file name, or click on the file name and choose OK.

Importing a DXF File

FlashCut provides a very useful 2D DXF import feature. The DXF import automatically arranges all lines and circles that have common endpoints into features. The entire toolpath is then optimized to reduce total machining time for the part. The DXF import assumes the part surface is at a Z program height of 0.0 and all geometry is to be treated as either holes or cutter paths with no offsets.

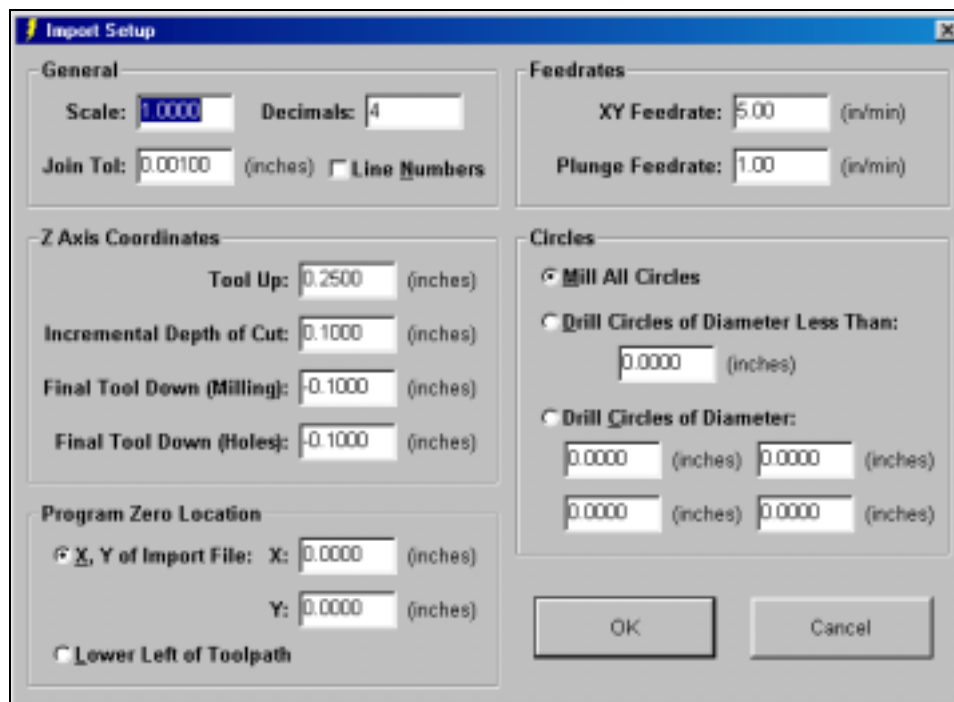
► To import a DXF file

1. Choose Import DXF from the File menu. The Import DXF File dialog box appears.
2. In the “List files of type” pull-down menu, choose the type of file you are looking for. A DXF file should have a “.DXF” extension.

3. In the Drives pull-down menu choose the drive that contains the file.
4. In Folders list box, double-click the name of the folder that contains the file. Continue double-clicking subfolders until you open the subfolder that contains the file.
5. In the box that lists files, double-click the file name, or click on the file name and choose OK.
6. The Save G-Code File dialog box appears asking you the name of the new G-Code file created by the DXF import. By default, it will use the DXF file name with an “.FGC” extension in the same drive and folder in which the DXF file resides. If this is acceptable, choose OK.

► **To choose a different file name or folder**

1. Choose the drive you want from the Drives pull-down menu.
2. Click on the folder you want in the Folder box.
3. Type the file name in the File name box, or type the entire path in the File name box.
4. Choose OK.
5. The Import Setup dialog box will appear.



6. Fill in the values for each of the following fields:
7. Scale - A multiplication factor for the toolpath in the XY plane only. For example, if you enter a 2 here, the toolpath generated will be double the size of the original geometry defined in the DXF file. Note that the values you enter for positioning the Z axis are unaffected by the scale factor.

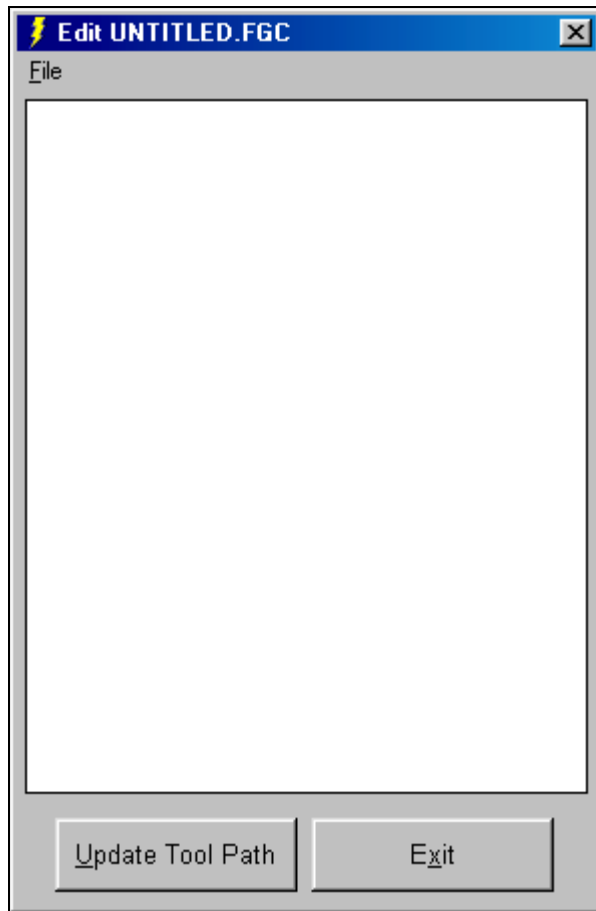
8. Decimals - The number of decimal places to use for all coordinates. A higher number can help eliminate extra backlash compensation moves caused by rounding error.
9. Join Tolerance - If two drawing entities, such as two lines, are touching end to end, FlashCut treats them as a single feature to machine without lifting the tool. Due to rounding or drawing error, two entities that are meant to be joined may not actually touch end to end. The DXF import will automatically join any entities whose endpoints are less than the Join Tolerance apart.
10. Line Numbers - Check this box if you want the DXF import to number all of the G-Code lines in the program it creates.
11. Tool Up - The height (program coordinates) to which the tool will move before rapid moves between two features.
12. Incremental Depth of Cut - The incremental depth for each milling pass. For example if the final tool down is -0.2500" and the incremental depth of cut is 0.0625" then four passes would be cut on each feature to get to the final depth of cut. (-0.0625, -0.1250, -0.1875, -0.2500). If the final tool down is -0.3000" and the incremental depth of cut is 0.0625" then five passes would be cut on each feature to get to the final depth of cut (-0.0625, -0.1250, -0.1875, -0.2500, -0.3000).
13. Final Tool Down (Milling) - The final depth (Program coordinates) to which the tool will cut each feature.
14. Final Tool Down (Holes) - The final depth (Program coordinates) to which the tool will cut each hole.
15. Program Zero Location - Determines where FlashCut will place Program Zero relative to the coordinate system of the import file. There are two options:
 - X, Y of Import File - You specify the X and Y location in the DXF file's coordinate system that FlashCut will place at Program Zero in the G-Code file.
 - Lower Left of Toolpath - FlashCut sets Program Zero at the lower left corner of the imaginary box that envelopes all geometry contained in the DXF file.
16. XY Feedrate - The feedrate for cutting moves in the XY plane.
17. Plunge Feedrate - The feedrate for all downward Z axis moves.
18. Circles - Defines diameters for circles that will be drilled (at the center point) instead of milled along the perimeter.
19. Choose OK.
20. FlashCut generates the G-Code file, displays it in the Program Listing Box, and displays the toolpath in the Toolpath View Port.

Using the Program Editor

FlashCut provides a simple editor for creating or modifying G-Code Programs. You can also use your own editor such as WordPad (which comes standard with Windows 95/98), Microsoft Word, and so on. If you use a different editor, make sure you save the file as Text Only and use an “.FGC” extension in the file name.

► To open the editor

1. Choose Editor from the File menu, or double-click the Program Listing Box. The editor dialog box will appear.



2. Type or edit your program in the scrolling text box. Refer to the “G and M Code Reference” section below to learn more about using G-Code.

► To edit a new program

1. Choose New Program from the editor’s File menu.

► To open an existing program

1. Choose Open G-Code from the editor’s File menu. The Open G-Code File dialog box appears.

2. In the “List files of type” pull-down menu, choose the type of file you are looking for. Existing FlashCut files will have an “.FGC” extension. If you are unsure of the file type, choose “All Files (*.*)”.
3. In the Drives pull-down menu choose the drive that contains the file.
4. In Folders list box, double-click the name of the folder that contains the file. Continue double-clicking subfolders until you open the subfolder that contains the file.
5. In the box that lists files, double-click the file name, or click on the file name and choose OK.

▶ **To see your new toolpath updated in the Toolpath View Port without leaving the editor**

1. Choose Update Tool Path.

▶ **To copy and paste text within the editor, or between the editor and other programs**

1. Use the standard Windows keyboard shortcuts as follows:

Copy	Ctrl + C
Paste	Ctrl + V

▶ **To save your program using the same file name**

1. Choose Save G-Code from the editor’s File menu.
2. If you’re saving a new file, a Save G-Code File dialog box will appear. Follow the directions “To save a new program...” below starting with step 2.

▶ **To save a new program created by the editor, or to save an edited file under a different file name**

1. Choose Save G-Code As from the editor’s File menu.
2. Choose the drive you want from the Drives pull-down menu.
3. Click on the folder you want in the Folder box.
4. Type the file name in the File name box, or type the entire path in the File name box.
5. Choose OK.

▶ **To close the editor**

1. Choose Exit.

G Codes Supported

G00	Rapid Tool Positioning
G01	Linear Interpolated Feedrate Move
G02	Clockwise Circular Feedrate Move
G03	Counter Clockwise Circular Feedrate Move
G04	Dwell
G17	XY Plane Selection
G18	XZ Plane Selection
G19	YZ Plane Selection
G20	Inch Units (same as G70)
G21	Metric Units (same as G71)
G28	Return to Reference Point
G29	Return from Reference Point
G43	Tool Length Compensation (Plus)
G44	Tool Length Compensation (Minus)
G49	Cancel Tool Length Compensation
G52	Use Local Coordinate System
G70	Inch Units (same as G20)
G71	Metric Units (same as G21)
G73	High Speed Peck Drilling Cycle
G80	Cancel Canned Cycle
G81	Drilling Cycle
G83	Peck Drilling Cycle
G90	Absolute Positioning Mode
G91	Incremental Positioning Mode
G98	Return to Initial Level (Canned Cycles)
G99	Return to Rapid Level (Canned Cycles)

M Codes Supported

M00	Program Pause
M02	End of Program
M03	Spindle On
M05	Spindle Off
M06	Tool Change
M07	Mist Coolant On
M08	Flood Coolant On
M09	Coolant Off
MXX	Custom Programmable (See “Output Line Settings” in the Software Setup section of this manual)
M30	End of Program (Reset)
M98	Subroutine Call
M99	Return From Subroutine
M100	Wait for Input Line (Normal State)
M101	Wait for Input Line (Tripped State)

Other Commands Supported

F	Feedrate
()	Comment

Key Programming Concepts

There are two basic programming concepts you should understand before learning the G and M codes - Mode and Absolute vs. Incremental.

Mode

Most G-code commands supported by FlashCut are **modal**, meaning they put the system into a particular mode of operation and need not be repeated on every program line. A modal command stays in effect until another command changes the mode. Related modal commands that affect one aspect of program execution are called a mode group.

The following list shows the mode groups supported by FlashCut.

Movement

- G00 Rapid Tool Positioning
- G01 Linear Interpolated Feedrate Move
- G02 Clockwise Circular Feedrate Move
- G03 Counter Clockwise Circular Feedrate Move

Circular Interpolation

- G17 XY Plane Selection
- G18 XZ Plane Selection
- G19 YZ Plane Selection

Units

- G20 Inch Units (also G70)
- G21 Metric Units (also G71)

Tool Length Compensation

- G43 Tool Length Compensation (Plus)
- G44 Tool Length Compensation (Minus)
- G49 Cancel Tool Length Compensation

Canned Cycles

- G73 High Speed Peck Drilling Cycle
- G80 Cancel Canned Cycle
- G81 Drilling Cycle
- G83 Peck Drilling Cycle

Canned Cycle Return

- G98 Return to Initial Level
- G99 Return to Rapid Level

Positioning

- G90 Absolute Positioning Mode
- G91 Incremental Positioning Mode

Independent Modal Commands

- G52 Use Local Coordinate System
- F Feedrate

Absolute vs. Incremental

All moves are either absolute or incremental. In an absolute move, the ending point is defined relative to Program Zero. In an incremental move, the ending point is defined relative to the current tool location. The G90/G91 commands tell the system which of these two modes to use (described below).

While there will be cases where incremental programming is useful, generally you should define your moves as absolute since it is a less error prone method of programming. All of the examples in the following section use absolute positioning unless otherwise noted.

G and M Code Reference

G00 Rapid Tool Positioning

The G00 command moves the tool to the designated coordinate at the rapid rate using 3-axis linear interpolation. The rapid rate is calculated from the Maximum Feedrates defined in the Feedrate/Ramping Setup dialog box.

Example:

G00 X1.0 Y2.0 Z1.5	Moves the tool to Program coordinates X=1.0, Y=2.0, Z=1.5 at the rapid rate
--------------------	---

When using G00, there are several things to keep in mind:

- While FlashCut supports machine tools with up to four axes, you may specify at most three coordinates for a G00 move. If you need to move all four axes, use two G00 commands.
- You do not need to specify three coordinates, only the ones for which you want movement.

Example:

G00 X4.0 Y3.0	Moves the tool to Program coordinates X=4.0, Y=3.0, leaving the Z position unchanged
---------------	--

- This is a modal command, meaning that all successive moves will be treated as rapid moves until another modal move command (G01, G02 or G03) occurs.

Example:

G00 X1.0 Y2.0 Z1.5	Rapid Move
X4.0 Y6.5 Z1.0	Rapid Move
G01 X3.0 Y3.0 Z1.4	Feedrate Move
X2.8 Y1.4 Z0	Feedrate Move

- The interpretation of the coordinates depends on the G90/G91 command in effect.

G01 Linear Interpolated Feedrate Move

The G01 command moves the tool to the designated Program coordinate at the specified feedrate using 3-axis linear interpolation.

Example:

G01 X2.0 Y1.0 Z-1.5 F2.0 Moves the tool to Program coordinates
X=2.0, Y=1.0, Z=-1.5 at a feedrate of 2.0
in/min

When using G01, there are several things to keep in mind:

- While FlashCut supports machine tools with up to four axes, you may specify at most three coordinates for a G01 move.
- You do not need to specify three coordinates, only the ones for which you want movement.

Example:

G01 X4.0 Y3.0 F2.0 Moves the tool to Program
coordinates X=4.0, Y=3.0, leaving
the Z position unchanged

- This is a modal command, meaning that all successive moves will be treated as linear feedrate moves until another modal move command (G00, G02 or G03) occurs.
- The interpretation of the coordinates depends on the G90/G91 command in effect.
- The F command is used to designate a feedrate. The feedrate set with the F command is modal (stays in effect until another F command occurs).

Example:

G01 X4.0 Y3.0 Z1.0 F7.0 Moves the tool to Program
coordinates X=4.0, Y=3.0, Z=1.0 at
a feedrate of 7.0 in/min

X2.0 Y2.5 Moves the tool to Program
coordinates X=2.0 Y=2.5, leaving
the Z axis unchanged at Z=1.0 (the
feedrate remains 7.0 in/min)

- For any move that includes the A (rotary) axis, the A axis always controls the timing. The F parameter is interpreted as degrees/minute.

Example:

G01 X1 A90 F360

The A axis turns 90 degrees while the X axis moves 1 inch. The A axis turns at exactly 360 degrees/minute. The timing for the X axis is

based on the A axis motion and is not explicitly listed in the G-code file.

Note that the F feedrate in use is always interpreted as degrees/minute for any move that contains motion on the A axis. For all other moves, it's interpreted as inches/minute (or mm/min). Therefore, you must be very careful to reset F every time you switch between pure linear and rotary-included moves.

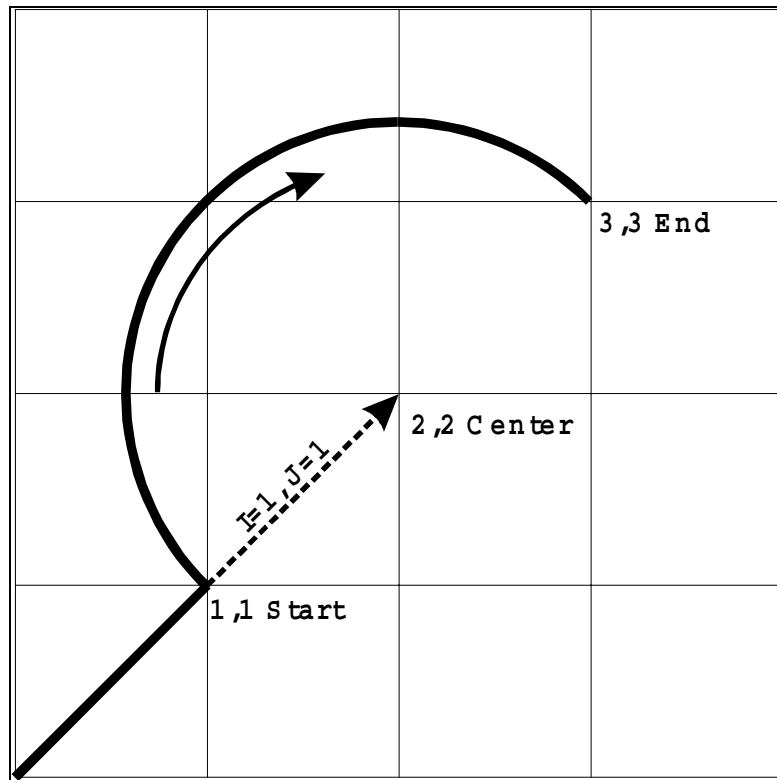
G02 Clockwise Circular Feedrate Move

The G02 command moves the tool in a clockwise path from the starting point (the current tool position) to the designated ending point in the currently selected plane (see G17-G19). The I, J, and K parameters represent the relative X, Y, and Z distances (respectively) from the starting point of the arc to the center point of the arc.

Example:

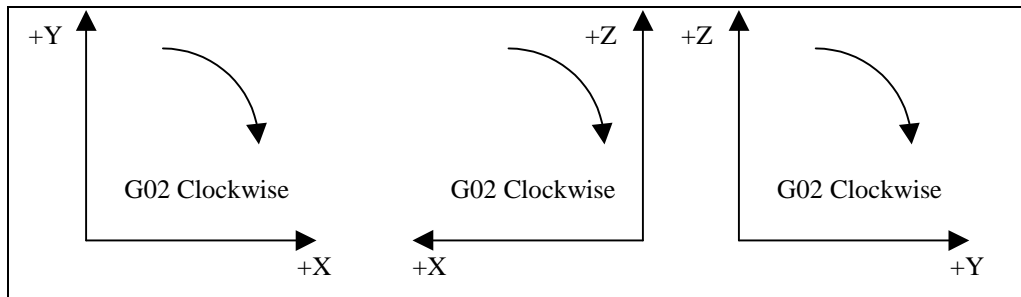
G01 X1.0 Y1.0 F3.0 Moves the tool to Program coordinates X=1.0, Y=1.0 at a feedrate of 3.0 in/min

G02 X3.0 Y3.0 I1.0 J1.0 Moves the tool using clockwise circular interpolation to Program coordinates X=3.0, Y=3.0 with a center point of X=2.0, Y=2.0 at a feedrate of 3.0 in/min



When using G02, there are several things to keep in mind:

- This is a modal command, meaning that all successive moves will be treated as clockwise circular feedrate moves until another modal move command (G00, G01 or G03) occurs.
- The interpretation of the X, Y and Z coordinates depends on the G90/G91 command in effect. The I, J and K values are unaffected by G90/G91.
- The tool will move at the current feedrate set by the last F command.
- Only XY arcs can be cut when G17 is active, only XZ arcs can be cut when G18 is active and only YZ arcs can be cut when G19 is active. Arcs cannot be specified for the A or W axes.
- The clockwise direction of rotation is as viewed from the positive end of the unused axis (the axis not in the plane of motion). For example, a G02 arc move in the XY plane is clockwise as viewed from the positive end of the Z axis (i.e. from above). The following diagram illustrates this behavior:

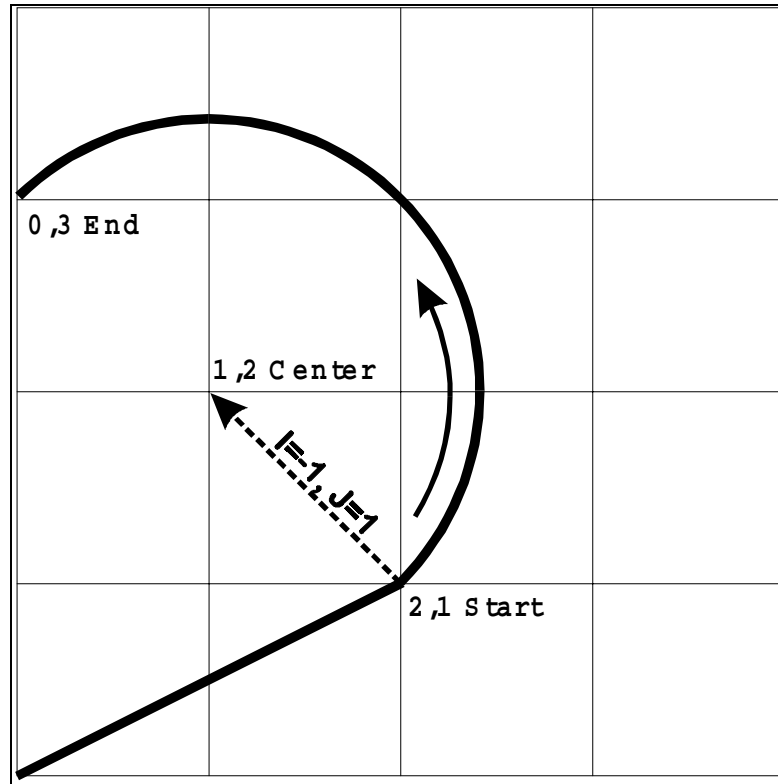


G03 Counter Clockwise Circular Feedrate Move

The G03 command is identical to the G02 command, but it moves the tool in a counter clockwise arc instead of a clockwise arc.

Example:

- | | |
|--------------------------|---|
| G01 X2.0 Y1.0 F8.0 | Moves the tool to Program coordinates X=2.0, Y=1.0 at a feedrate of 8.0 in/min |
| G03 X0.0 Y3.0 I-1.0 J1.0 | Moves the tool using counter clockwise circular interpolation to the Program coordinates X=0.0, Y=3.0 with a center point of X=1.0, Y=2.0 at a feedrate of 8.0 in/min |



G04 Dwell

The G04 command causes the program to dwell for a specified amount of time. The command format is:

G04 Xn

where n is the number of seconds to wait. For safety reasons there is a maximum time allowed for each dwell command.

Example:

G04 X1.5

Program pauses for 1.5 seconds before moving on to the next line of G-Code

G17, G18, G19 Arc Plane Selection

These commands specify the plane used for circular interpolation as follows:

G17 XY plane

G18 XZ plane

G19 YZ plane

When using G17-G19, there are several things to keep in mind:

- Unless you explicitly use the G18 or G19 command, FlashCut assumes G17 as the default.

- The three commands are modal, i.e. one command remains in effect until another in the set is used.

G20, G21 Inch Units and Metric Units

The G20 command indicates that all G-Code commands are in inch units. FlashCut then assumes all distances are in inches and all feedrates are in inches/minute. For compatibility reasons, FlashCut accepts G70 as equivalent to G20.

The G21 command indicates that all G-Code commands are in metric units. FlashCut then assumes all distances are in millimeters and all feedrates are in millimeters/minute. For compatibility reasons, FlashCut accepts G71 as equivalent to G21.

When using G20 or G21, there are several things to keep in mind:

- If you don't use either command, FlashCut assumes all program values are consistent with the Display Units setting in the System Options dialog box.
- You may only use one of these two commands in any G-Code program, so all values in a G-Code file must use the same unit system.

G28 Return to Reference Point

The G28 command moves the tool at the rapid rate to the Tool Change Position defined in the Machine Tool Setup dialog box. This position is defined in Machine Coordinates, so Machine Zero must be set for this command to be used.

The sequence of axis motions follows a general-purpose scheme that's described under "Point Control Panel" in the Main Screen Features section of this manual.

If you want the G28 command to move only some of the axes on your machine tool, you can limit the movement to those axes by adding the parameters "X0", "Y0", "Z0" or "A0" after the G28 command. Then, FlashCut moves only the indicated axes to their Tool Change Position coordinates. A typical use is to only raise the Z axis for a manual tool change ("G28 Z0").

Example:

The Tool Change Position is defined as Machine Coordinate X=1, Y=1, Z=-1.

G01 X1.5 Y2.5 Z-3 F10	Linear move to the Program coordinates X=1.5, Y=2.5, Z=-3
G28 Z0	Rapid move in the Z axis only to Machine coordinate Z=-1
G01 X1.5 Y2.5 Z-3 F10	Linear move to the Program coordinates X=1.5, Y=2.5, Z=-3

G28	Rapid move in the Z axis to Machine coordinate Z=-1 followed by a rapid move in the XY plane to Machine coordinates X=1, Y=1
G01 X1.5 Y2.5 Z-3 F10	Linear move to the Program coordinates X=1.5, Y=2.5, Z=-3
G28 X0 Y0 Z0	Rapid move in the Z axis to Machine coordinate Z=-1 followed by a rapid move in the XY plane to Machine coordinates X=1, Y=1 (identical to the G28 command with no parameters specified)

G29 Return from Reference Point

The G29 command moves the tool to the designated XYZA coordinate at the rapid rate.

The sequence of axis motions follows a general-purpose scheme that's described under "Point Control Panel" in the Main Screen Features section of this manual.

Example:

The Tool Change Position is defined as Machine Coordinate X=1, Y=1, Z=-1.

G01 X1.5 Y2.5 Z-3 F10	Linear move to the Program coordinates X=1.5, Y=2.5, Z=-3
G28	Rapid move in the Z axis to Machine coordinate Z=-1 followed by a rapid move in the XY plane to Machine coordinates X=1, Y=1
G29 X2 Y3 Z-2	Rapid move in the XY plane to Program coordinates X=2, Y=3 followed by a rapid move in the Z axis to Program coordinate Z=-2

When using G29, there are several things to keep in mind:

- You do not need to specify coordinates for all machine tool axes, only the ones for which you want movement.

Example:

G29 X4.0 Y3.0	Moves the tool to Program coordinates X=4.0, Y=3.0, leaving the Z position unchanged
---------------	--

- The interpretation of the coordinates depends on the G90/G91 command in effect.

G43, G44, G49, M06 Tool Change and Tool Length Compensation Commands

FlashCut supports tool changes and tool length compensation. Tool length compensation lets FlashCut account for differences in tool lengths, so the G-Code program can be created without regard to specific tool lengths (except for possible interference problems).

THESE COMMANDS ARE NOT FOR THE NOVICE CNC USER. WHEN NOT PROPERLY USED, TOOL LENGTH COMPENSATION CAN CRASH THE MACHINE TOOL, CAUSING SERIOUS DAMAGE TO YOUR WORKPIECE OR MACHINE TOOL.

When applying tool length compensation, FlashCut uses the Length Offsets defined in the Tooling Setup dialog box. See “Tooling Settings” in the Initial Setup section for more information on defining your tools.

For tool length compensation to work properly, FlashCut must know what tool is in use at all times, including the tool that’s loaded when you start running a G-Code file. Therefore, before you run a G-Code file that uses tool length compensation, you must first choose your starting tool from the Current Tool pull-down menu on the main screen.

To indicate tool changes in the G-Code file, use the M06 command as follows:

`M06 Tn`

where n is the tool number in the Tooling Setup dialog box.

Example:

<code>M06 T3</code>	Pauses program, displays dialog informing operator to change to tool number 3
---------------------	---

Note: For compatibility reasons, the T command can be used on any line prior to the M06 command; it does not need to be on the same line as M06.

Once the M06 command has set the current tool, the G43 command applies the proper offset to account for the current tool’s length as follows:

`G43 Hn`

where n is the tool number for the current tool.

The G43 command tells FlashCut to shift all subsequent Z axis moves away from the workpiece (in the positive Z direction) by an offset amount. The offset amount is equal to the difference in lengths between the current tool when the G43 command is executed and the previous tool. FlashCut uses the Length Offset values in the Tooling Setup dialog box to calculate the difference in lengths between two tools.

Example:

G43 H3	Shifts all subsequent Z axis moves away from the workpiece (in the positive Z direction) by the difference in lengths between tool number 3 and the previous tool
--------	---

The G44 command is identical to the G43 command, except that it shifts all Z axis moves in the direction opposite from G43. Unless you are an experienced CNC programmer and know how to use G44 correctly, G43 is the preferred command.

The G49 command cancels tool length compensation. It removes any offset that FlashCut has applied since the G-Code program began running.

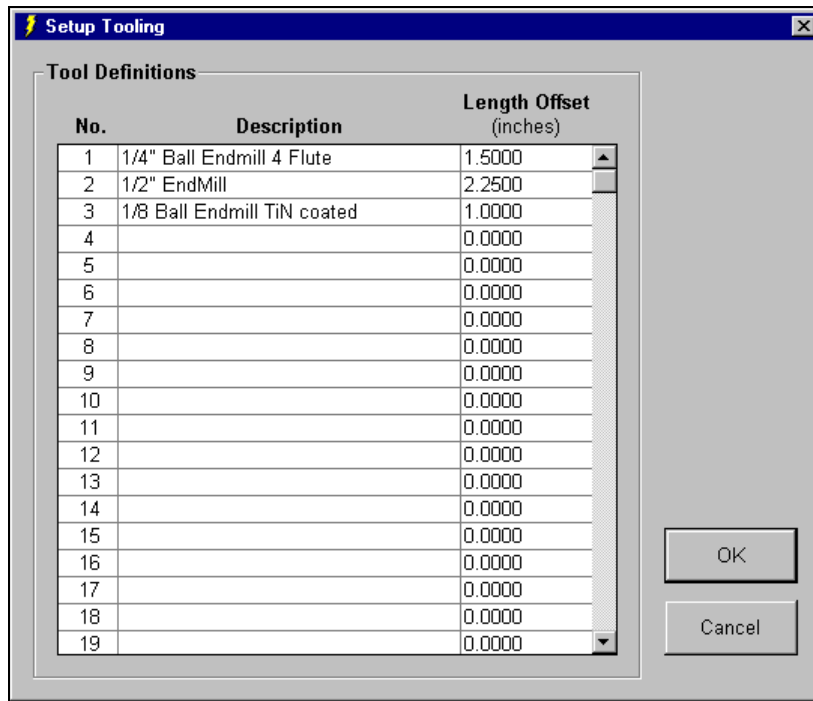
When using tool length compensation there are several important things to keep in mind:

- You must predefine all tools and tool lengths in the Tooling Setup dialog box.
- It's good practice to include an M06 tool change and a G43 compensation command for the first tool used, near the beginning of the G-Code program.
- FlashCut automatically cancels tool offset when it finishes processing a G-Code file, or during any operation that ends the current run of the G-Code file (such as resetting the program, opening a new program, and so on.) **TO AVOID CRASHING THE MACHINE TOOL, IT IS VERY IMPORTANT THAT YOU REMOVE THE CURRENT TOOL FROM THE SPINDLE WHENEVER FLASHCUT CNC CANCELS TOOL OFFSET.**
- The G43, G44 and G49 commands are modal, so the current tool offset remains active until FlashCut executes another tool offset command, or until FlashCut cancels tool offset as described above. Note that you may only use one type of tool length compensation (G43 or G44) in a G-Code program.
- The M06 command does not move the machine tool to the Tool Change Position. This is done using the G28 command described above. It's good practice to place the G28 command in the line directly preceding the M06 command.
- It's good practice to use the G43 command in the line directly following the M06 command.

The following example illustrates proper use of the tool change and tool length compensation commands.

Example:

The tool library is shown below.



The first tool used in the program is tool #1, so it is selected in the Current Tool pull-down menu on the main screen. The tool change position is defined as Machine Coordinates X=2, Y=2, Z=0. Tool #1 is loaded in the machine tool. Program zero has been set using tool #1. Program zero is set at Machine Coordinates X=0, Y=0, Z=-4. The machine tool is moved to Program Coordinates X=0, Y=0, Z=1 before the G-Code file is run.

M03	Turns on the spindle
G00 Z.25	Move the Z axis down to Program coordinates X=0, Y=0, Z=0.25, Machine coordinates X=0, Y=0, Z=-3.75
G01 Z-1.0 F10	Move the Z axis down to Program coordinates X=0, Y=0, Z=-1, Machine coordinates X=0, Y=0, Z=-5
G00 Z.25	Move the Z axis up to Program coordinates X=0, Y=0, Z=0.25, Machine coordinates X=0, Y=0, Z=-3.75
G28	Move the Z axis up and the X and Y axes across to the Tool Change Position, Program coordinates X=2, Y=2, Z=4, Machine coordinates X=2, Y=2, Z=0
M06 T2	Change the tool to Tool #2 (all coordinates remain unchanged, and the spindle is turned off)

G43 H2	Apply the tool length compensation for tool #2. The offset amount is 0.750 (2.250-1.500). The tool does not move. However, the Program coordinates change to X=2, Y=2, Z=3.25. The Machine coordinates remain unchanged at X=2, Y=2, Z=0
M03	Turns on the spindle
G29 X1 Y1 Z0.25	Move the X and Y axes across and the Z axis down to Program coordinates X=1, Y=1, Z=0.25, Machine coordinates X=1, Y=1, Z=-3.0
G01 X3 Y3 Z-1 F20	Linear interpolation to Program coordinates X=3, Y=3, Z=-1, Machine coordinates X=3, Y=3, Z=-4.25
G28 Z0	Rapid move in the Z axis only to Program coordinates X=3, Y=3, Z=3.25, Machine coordinates X=3, Y=3, Z=0
M06 T3	Change the tool to tool #3 (all coordinates remain unchanged, and the spindle is turned off)
G43 H3	Apply the tool length compensation for tool #3. The offset amount is -1.250 (1.000-2.250). The tool does not move. However, the Program coordinates change to X=3, Y=3, Z=4.5. The Machine coordinates remain unchanged at X=3, Y=3, Z=0
M03	Turns on the spindle
G29 X4 Y4 Z0	Move the X and Y axes across and the Z axis down to Program coordinates X=4, Y=4, Z=0, Machine coordinates X=4, Y=4, Z=-4.5
G01 X5 Z-1	Linear interpolation to Program coordinates X=5, Y=4, Z=-1, Machine coordinates X=5, Y=4, Z=-5.5
G28	Move the Z axis up and the X and Y axes across to the Tool Change Position, Program coordinates X=2, Y=2, Z=4.5, Machine coordinates X=2, Y=2, Z=0

G49 Cancel Tool Length Compensation. The Z axis Program coordinate changes by -0.500, the difference in Length Offset between the current tool (#3: 1.000) and the tool displayed in the Current Tool pull-down menu when the program first began (#1: 1.500). The new Program coordinates are X=2, Y=2, Z=4. The Machine coordinates remain unchanged at X=2, Y=2, Z=0. At this point the current tool should be removed from the spindle

G52 Local Coordinate System

The G52 command defines and activates a local coordinate system that FlashCut uses in place of your original Program coordinates for all absolute positioning moves. The X, Y, Z and A parameters indicate the offset from your original Program Zero location to the origin for the local coordinate system.

For example, "G52 X1 Y2 Z-4" would activate a local coordinate system whose origin is at a distance of 1, 2, -4 from the original Program Zero.

All absolute moves are made relative to the new local coordinate system. To cancel use of the local coordinate system in the middle of a G-code file, use the command "G52 X0 Y0 Z0 A0" (using only the letters for axes used on your machine).

When FlashCut reads a G52 command, it displays a magenta dot in the Toolpath View Port showing the origin of the local coordinate system.

Note that the local coordinate system only applies to the G-code file being executed. The G52 command has no effect on the Program Zero you set before running the G-code file. FlashCut automatically cancels the local coordinate system when it completes execution of a G-code file.

Example:

G01 X1.0 Y3.0 Z-1.5 F12	Moves the tool to Program coordinates X=1.0, Y=3.0, Z=-1.5
G52 X3 Y-7 Z0	Activates a local coordinate system with origin at X=3, Y=-7, Z=0 relative to Program Zero (the machine tool does not move)
G01 X1.0 Y10.0 Z2.0	Moves the tool to the point X=1.0, Y=10.0, Z=2.0 relative to the local coordinate system as defined by the G52 command above

G52 X0 Y0 Z0

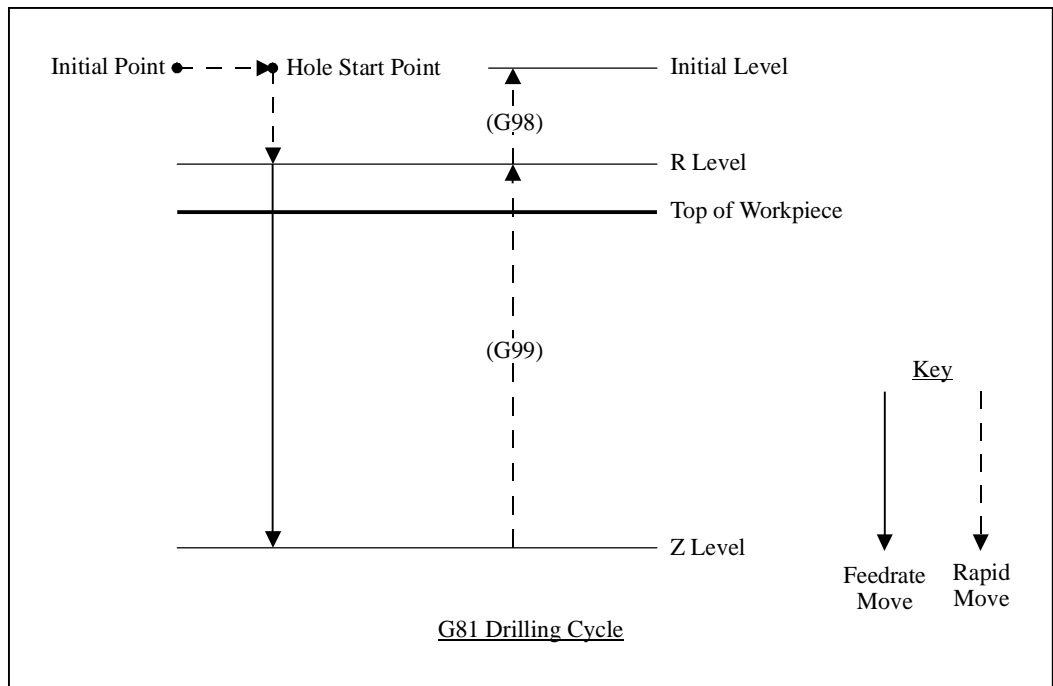
Cancels use of the local coordinate system. All absolute moves are again relative to Program Zero as you set it before running the program

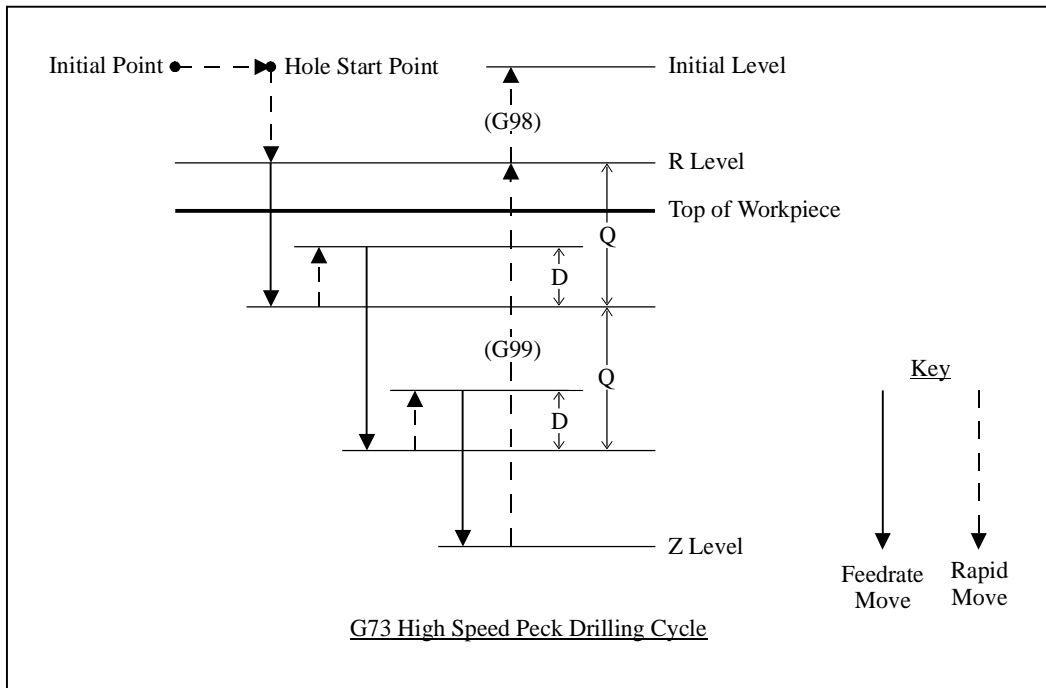
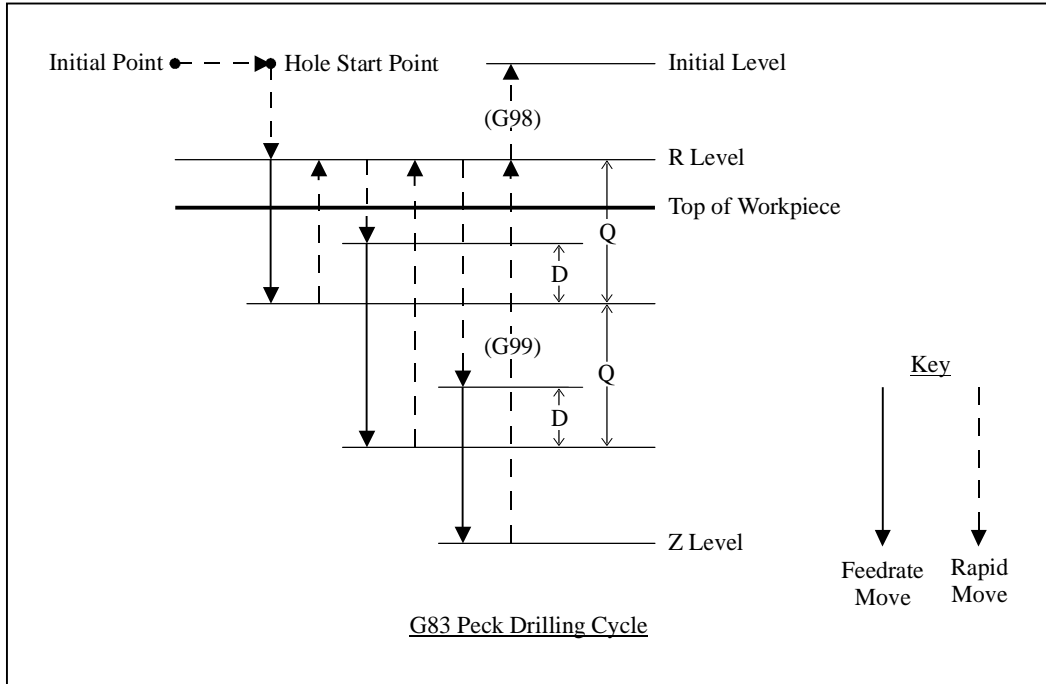
G73, G80, G81, G83, G98, G99 Canned Cycle Commands

Canned cycles simplify programming by letting you specify standard machining operations with a single command. FlashCut currently supports the following canned cycle commands:

- G73 High Speed Peck Drilling Cycle
- G80 Cancel Canned Cycle
- G81 Drilling Cycle
- G83 Peck Drilling Cycle

The following diagrams illustrate the sequence of moves for the three drilling cycles. The rest of this section refers to these diagrams.





The general command syntax is as follows:

Gn Xx Yy Rr Zz Qq Ff Ll

where the parameters are

- n: Canned cycle number (73, 81 or 83)
- x: X coordinate of Hole Start Point (G90 mode) or distance for rapid move from Initial Point to Hole Start Point (G91 mode)
- y: Y coordinate of Hole Start Point (G90 mode) or distance for rapid move from Initial Point to Hole Start Point (G91 mode)
- r: Z coordinate of R Level (G90 mode) or distance for rapid move from Hole Start Point to R Level (G91 mode)
- z: Z coordinate of Z Level (G90 mode) or distance from R Level to Z Level (G91 mode)
- q: Depth of cut for each downward cutting move (G73/G83 only)
- f: Feedrate for each downward cutting move
- l: Number of times to repeat the canned cycle (G91 mode only)

The required parameters are R, Z, F and Q (G73/G83 only).

The G98 and G99 commands affect the final Z axis rapid move, upward out of the completed hole (rightmost move in each diagram), as follows:

G98: Z axis moves up to the Initial Level

G99: Z axis moves up to the R Level

G98 and G99 compose a mode group, with G98 being the default if neither is specified in the G-Code file. You can include either command on the same line as the canned cycle command (before or after the canned cycle command and all parameters).

The D distance is specified in the G73/G83 Retract Distance field in the G/M Code Setup dialog box.

Example (assumes D is 0.050 in.):

G98	Optional
G00 X0.0 Y0.0 Z1.0	Moves the tool to the Initial Point, Program coordinate X=0.0, Y=0.0, Z=1.0 (not required, for illustration only)

G83 X1.0 Y2.0 R0.1 Z-1.0 Q0.5 F8.0

First moves the tool to the Hole Start Point, Program coordinates X=1.0, Y=2.0, Z=1.0. Then peck drills a 1.0" deep hole (rapid down to 0.1, feedrate down to -0.4, rapid up to 0.1, rapid down to -0.350, feedrate down to -0.9, rapid up to 0.1, rapid down to -0.850, feedrate down to -1.0, rapid up to 1.0)

When using the canned cycle commands, there are several things to keep in mind:

- G73, G80, G81 and G83 compose a mode group.
- G80 cancels the active canned cycle command. The G00, G01, G02 and G03 commands also cancel the active canned cycle command.
- While a canned cycle command is active, the command and its R, Z, Q, F, and L parameters do not need to be repeated on every G-Code line. However, the X and/or Y positioning parameters must be included, and must start the G-code line for the line to be interpreted as a canned cycle command.

Example (drills 3 holes in a row along the X axis using the G81 cycle):

```
G81 X1.0 Y2.0 R0.25 Z-0.5 F5
X2.0
X3.0
```

- When you switch to a new command in the mode group, you must include all parameters required for the command.
- The feedrate set by the F parameter for any canned cycle command remains as the current feedrate for subsequent G01, G02 or G03 moves, unless you explicitly set F to another value for the next move.

G90 Absolute Positioning Mode

The G90 command puts the system into absolute positioning mode. All XYZA coordinates are treated as points relative to Program Zero (or a local coordinate system set by the G52 command). This command stays in effect until a G91 command occurs.

Note that absolute positioning is the default positioning mode for FlashCut. It is not necessary to include this command in your G-code file if all your moves are absolute.

G91 Incremental Positioning Mode

The G91 command puts the system into incremental positioning mode. All XYZA coordinates are treated as incremental move distances. This command stays in effect until a G90 command occurs.

Example:

G01 X1.0 Y3.0 Z-1.5 F12	Moves the tool to Program coordinates X=1.0, Y=3.0, Z=-1.5 (G90 is assumed)
G91	All XYZA coordinates after this command will be interpreted as incremental distances.
G01 X1.0 Y2.0 Z-0.5	Moves the tool a distance of X=1.0, Y=2.0, Z=-0.5 from the current tool location (this corresponds to the Program coordinates X=2.0, Y=5.0 Z=-2.0)
G02 X1.0 Y-1.0 I0.5 J-0.5	Moves the tool using counter-clockwise circular interpolation to the Program coordinates X=3.0, Y=4.0, Z=-2.0 with a center point at Program coordinates X=2.5, Y=4.5, Z=-2.0.
G90	All XYZA coordinates after this command will be interpreted as Program coordinates
G01 X1.0 Y2.0 Z-0.5	Moves the tool to Program coordinates X=1.0, Y=2.0, Z=-0.5

M00 Program Pause

The M00 command pauses processing of the G-Code program. By default, FlashCut displays a dialog box to inform you that it has paused processing. You can control whether or not this dialog box appears using the Message on M00 Program Pause checkbox in the G/M Code Setup dialog box.

M30 End of Program

The M30 command ends processing of the G-Code program and automatically resets the program to the first line.

M98, M99, M02 Subroutine Commands

Subroutines allow you to eliminate repetitive programming. FlashCut supports the use of subroutines with the M98, M99, and M02 (or M30) commands. Use of these commands is best explained through a simple example. The following G-code program uses one subroutine called "mysub":

Example:

G01 X1 Y1 F10	First line of main program
M98 Pmysub	Jump to subroutine "mysub"
G01 X0 Y0	Continued execution after "mysub" ends
M02	End of main program
Omysub	First line of the subroutine called "mysub"
G01 X2 Y2	Continued execution within the subroutine
M99	End of subroutine "mysub"

In the main program, the M98 command causes program execution to jump to the first line of the subroutine named "mysub". Notice that the letter "P" must immediately precede the name of the subroutine with no spaces.

The subroutine definition begins with the letter "O" followed immediately by the subroutine name with no spaces. The subroutine must end with the M99 command as shown. M99 causes program execution to jump back to the main program, continuing with the line immediately following the M98 line (G01 X0 Y0 above).

When subroutines are used, the main program must end with M02 or M30, the "End of Program" commands. Subroutine names may include up to 10 alphanumeric characters. You can "nest" subroutines as much as you like, meaning one subroutine may call another subroutine, which in turn may call another subroutine, and so on.

To call the same subroutine multiple times, include the L parameter as shown below.

Example:

M98 Pmysub L10	Execute subroutine "mysub" 10 times
----------------	-------------------------------------

This feature provides a simple means to repetitively loop through any block of G-Code. Just place the G-Code in a subroutine, then call the subroutine with the L parameter set to the number of loops required.

It's sometimes convenient to place subroutines in files separate from the main G-Code file. These files are called **included files**, and you may use as many as you like. You must list each included file at the top of the main G-Code file using the following syntax:

```
#[file path and name]
```

Once a file has been included in the main G-Code file, FlashCut treats the included file's subroutines the same as if they were listed at the bottom of the main G-Code file. If you leave out the path, FlashCut will look for the file in the same folder as the main G-Code file.

Example:

Main G-Code File (Engraves “A B”)

```
#C:\GCODE\ALPHABET.FGC
```

```
G00 X0 Y0
```

```
M98 PmakeA
```

```
G00 X1 Y0
```

```
M98 PmakeB
```

```
M02
```

ALPHABET.FGC File

```
OmakeA
```

```
G91
```

```
[Incremental G-Code to cut the letter “A”]
```

```
G90
```

```
M99
```

```
OmakeB
```

```
G91
```

```
[Incremental G-Code to cut the letter “B”]
```

```
G90
```

```
M99
```

Notice that the syntax for calling the included subroutines is the same as for calling subroutines listed in the main G-Code file.

You may create as many subroutines as you like in the main G-Code file and all included files, but each must have a unique name.

Flashcut’s support for included files lets you create a library of reusable subroutines that you can apply to various projects and jobs.

M100, M101 Wait for Input Line

The M100 and M101 commands wait for an input line to reach a desired state as follows:

M100 Wait for normal state

M101 Wait for tripped state

The Signal Generator setting in the System Options dialog box determines the wiring required for these commands as shown below.

Signal Generator Setting	401A (Inputs Normally Closed)	401 (Inputs Normally Open)
M100 Waits For Input Line...	Closed	Open
M101 Waits For Input Line...	Open	Closed

The command syntax is as follows:

M100 Ix

M101 Ix

where x is the number of the input line to monitor.

Examples:

M100 I2 System waits for input line 2 to reach the normal state

M101 I7 System waits for input line 7 to become tripped

You must define the specified input line as “Control” in the Switch Function pull-down menu, in the Input Lines Setup dialog box (see “Input Line Settings” in the Initial Setup section for more details).

FlashCut uses debounce for these commands. For a switch to successfully reach a desired state, it must hold that state continuously for the debounce duration. You can set the M100/M101 Debounce time in the G/M Code Setup dialog box.

The M100 and M101 commands time out once the maximum allowable time has passed and the input line has not reached the desired state. When the commands time out, FlashCut halts processing of the G-Code program and displays a message to the operator. You can set the M100/M101 Timeout duration in the G/M Code Setup dialog box.

A typical use for these commands is detecting completed motion of an auxiliary device, such as an indexer.

M03, M05, M07, M08, M09, MXX Miscellaneous Device Control

Using the Output Lines Setup dialog box you can define up to 16 M codes to turn on or off different devices through the output lines. M Codes can also be used for digital control of devices by turning on or off a group of output lines to be used as digital input into the control lines of the device. See “Output Lines Settings” in the Initial Setup section for details on how to set up the M codes.

Typical M codes include:

M03 Spindle On

M05 Spindle Off

M07 Mist Coolant On

M08 Flood Coolant On

M09 Coolant Off

F Feedrate Command

The F command is used to designate a feedrate. The feedrate set with the F command is modal (stays in effect until another F command occurs).

For all linear moves, specify the feedrate in inches/minute for English units and millimeters/minute for Metric units.

Example:

```
G01 X4.0 Y3.0 Z1.0 F7.0    Moves the tool to Program coordinates
                           X=4.0, Y=3.0, Z=1.0 at a feedrate of 7.0
                           in/min
```

For any move that includes the A (rotary) axis, specify the feedrate in degrees/minute. Please see the detailed explanation of rotary feedrates under “G01 Linear Interpolated Feedrate Move” above.

Program Comments

You can add comments to your program by enclosing them in parentheses. FlashCut ignores anything enclosed in parentheses as shown below.

Example:

```
(Move to beginning of the next feature)
G00 X1.0 Y3.0 (Ready to move Z axis down)
G00 Z-1.5
(Begin next feature)
G01 Z-1.6 F8
G01 X3.0 Y7.5
```


Section 5. Tutorial

Starting FlashCut CNC

Windows 3.1 or 3.11

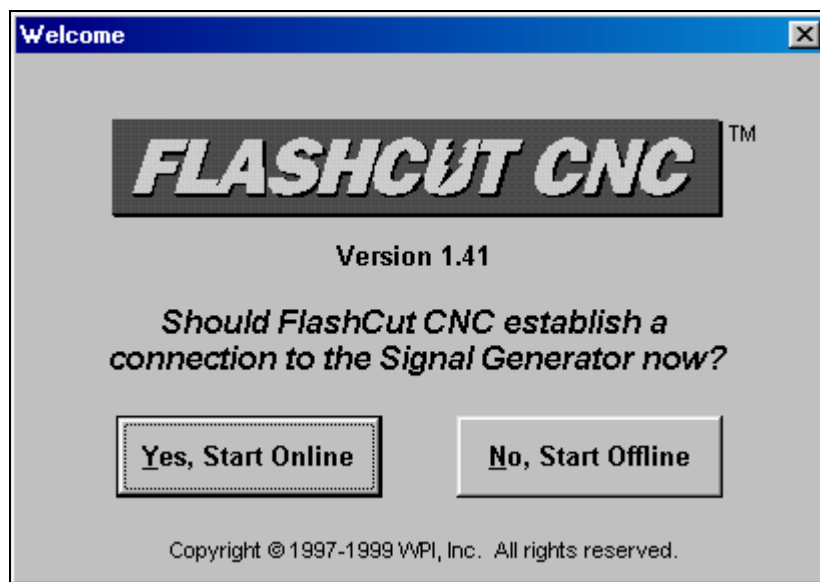
To start FlashCut CNC, double-click on the FlashCut CNC icon in the FlashCut CNC Program Group.

A dialog will appear asking you if you want to start with the Signal Generator online or offline. Choose the No, Start Offline button. If you're running the Demo version, choose Continue.

Windows 95, 98 or NT

To start FlashCut CNC, click on the Start button, select Programs, select FlashCut CNC, and then select the FlashCut CNC icon.

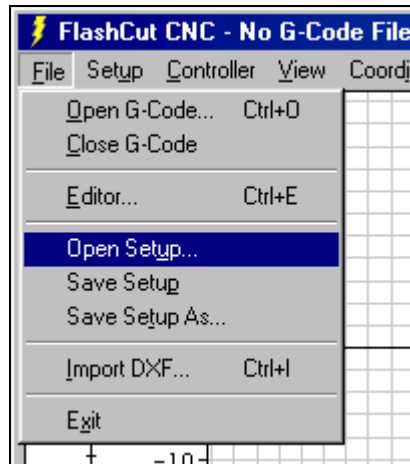
A dialog will appear asking you if you want to start with the Signal Generator Online. Choose the No, Start Offline button. If you're running the Demo version, choose Continue.



Configuring FlashCut CNC

1. Refer to the Initial Setup section of this manual to properly configure FlashCut and your Signal Generator. If you've already made a setup file, or want to use a predefined setup file do the following:

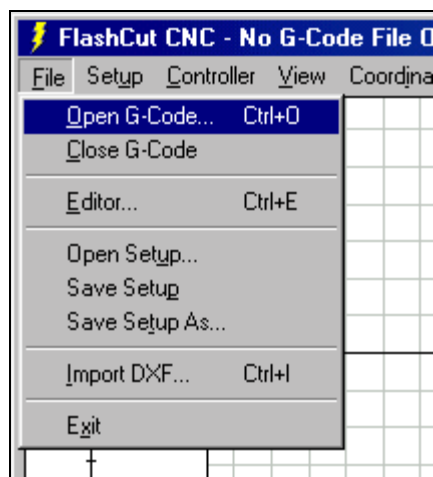
- Choose Open Setup from the File menu.



- Select the drive and directory where the setup file is located, then select the file and choose OK. Some setup files are supplied for various mills and lathes. If a setup file is not available for your machine, select FLASHXXX.STP, where "XXX" is the current software version (e.g. "FLASH141.STP" for version 1.41). You can easily configure the setup parameters in this file for your machine tool. If you're running the Demo version, you can use the default parameters.
- Go through the Setup dialog boxes as described in the Initial Setup section.
- Choose All Coordinates from the View menu to make sure all four coordinate systems are displayed.

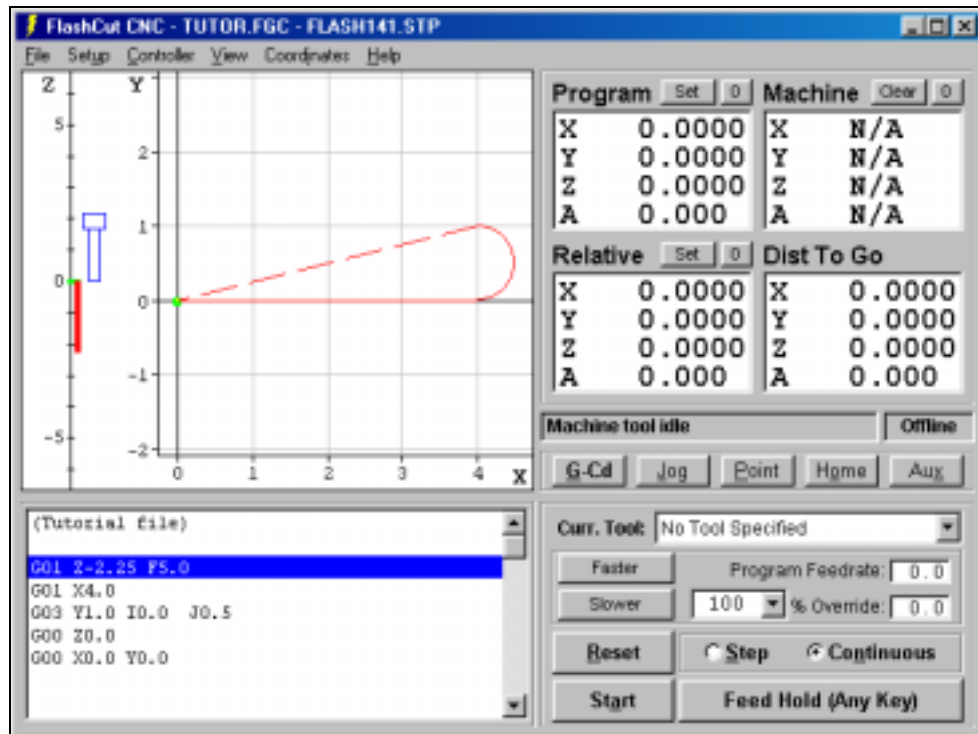
Loading a G-Code File

- Choose Open G-Code from the File menu.



- Go to the directory where FlashCut is installed and double-click on the file TUTOR.FGC.

FlashCut will load TUTOR.FGC and the screen should now look like this:



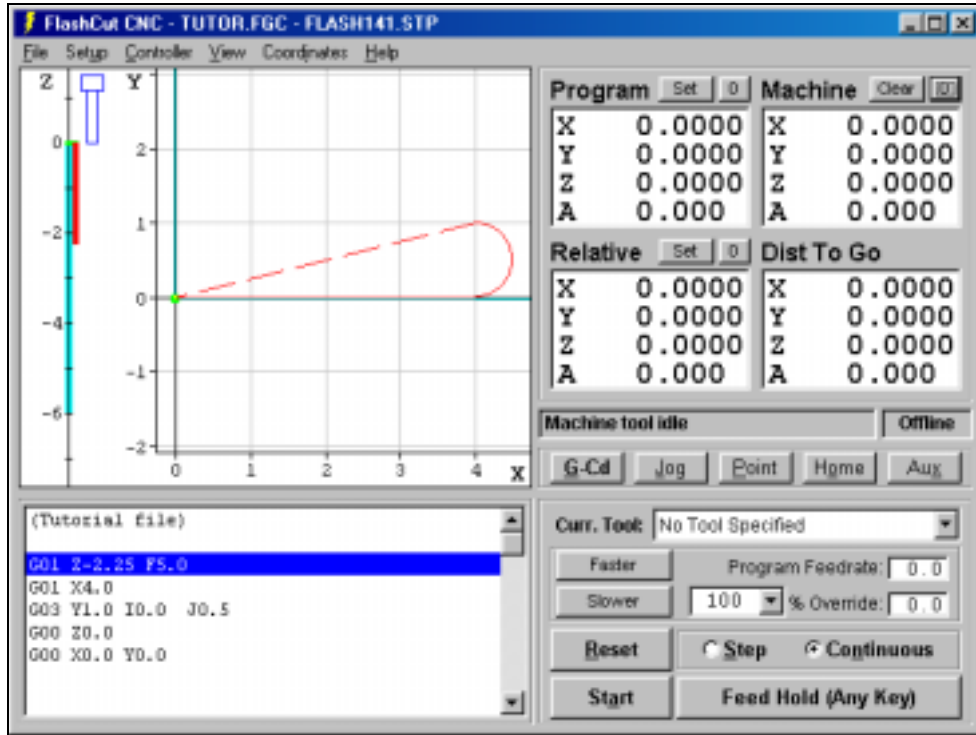
Notice how the G-Code listing appeared in the Program Listing Box and a red outline of the toolpath appeared in the Toolpath View Port.

Viewing the Tool Path

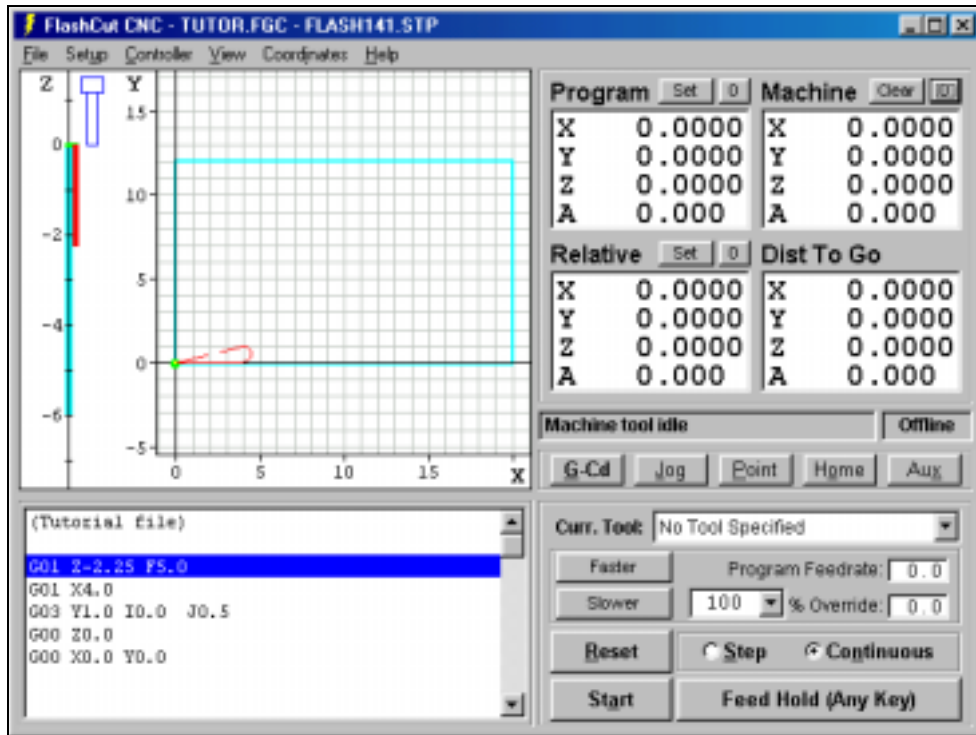
Before examining the toolpath display let's zero the Machine coordinates. To do this:

1. Choose the Zero (0) button to the right of the Machine label in the Tool Position Box. Choose Yes in the confirmation dialog that follows.

The Machine coordinates, previously shown as N/A, will now be zeroed and a light blue box will outline the tool envelope in the Toolpath View Port. Because Scale to Fit is checked in the View menu, you'll only see a portion of the tool envelope boundary, as shown below.



To see the entire machine tool envelope, choose Scale to Fit from the View menu, which removes the checkmark from the Scale to Fit command. The screen will then appear like this:



Choose Scale to Fit from the View menu again, to see the largest possible image of the toolpath.

Now let's get familiar with the Toolpath View Port. Here are some important features:

Red Lines - Represent the toolpath for the G-Code program.

Green Dot - Represents Program Zero, the origin of any G-Code program.

Light Blue Dot - Represents Machine Zero, also called Home.

Yellow Dot (not shown here) - Represents the current XY position of the machine tool during the cutting operation.

Dark Blue Lines (not shown here) - Represent the portion of the toolpath already cut.

Dotted Lines - Represent a rapid move.

Solid Lines - Represent a feedrate move.

Blue Tool Icon - Represents the current Z axis position (bottom of icon).

Light Blue Lines - Represent the borders of the machine tool envelope.

Animating the G-Code File

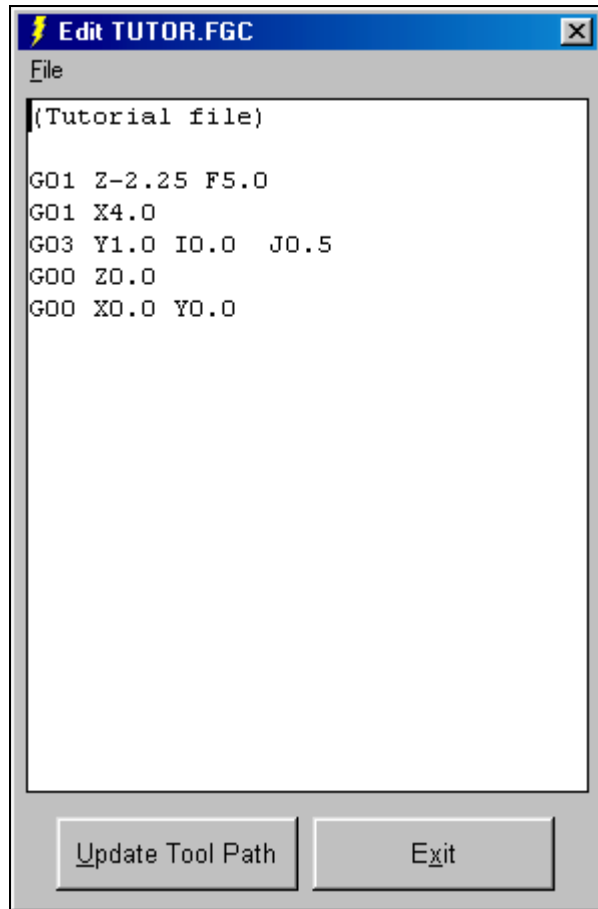
Now we're ready to animate and verify the toolpath on the screen.

1. Choose the Zero (0) button next to the Program label, then choose OK in the confirmation dialog box that follows. FlashCut sets all four Program coordinates to 0. This simulates the tool being in the correct position before the G-Code program begins.
2. Choose the G-Cd button in the Control Selection Box to make sure the G-Code Control Panel is displayed.
3. Select the Step radio button so FlashCut will execute the G-Code program one line at a time.
4. Choose the Start button and watch the blue tool move down the Z axis scale. Notice that FlashCut has highlighted the next line in the Program Listing Box, indicating it has fully executed the first line.
5. Choose the Start button again. Notice the yellow dot, which represents the current position of the tool, and the solid blue line, which represents the cutting move just executed.
6. Select the Continuous radio button, then choose the Start button again. The yellow dot quickly moves along the toolpath to completion, and the path is traced in blue as it's cut. At the end of the G-Code program, the computer will beep, indicating the file was successfully processed.

Editing a G-Code File

Now let's get familiar with using the FlashCut editor.

1. Choose Editor from the File menu. The editor dialog box will appear.



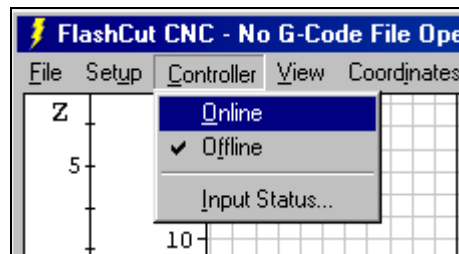
2. First, let's reduce the cutting depth from 2.25" to 0.25". With your mouse or your arrow keys, move the cursor to the first line of G-Code that reads: G01 Z-2.25 F5.0. Change it to read: G01 Z-0.25 F5.0. Then choose Update Tool Path at the bottom of the editor dialog box. Notice how the red line along the Z Scale, which shows the total length of Z travel, shrinks down to 0.25" long.
3. Now, let's change the diameter of the arc we're cutting. Move the cursor to the line of G-Code that reads: G03 Y1.0 I0.0 J0.5. Change it to read: G03 Y2.0 I0.0 J1.0.
4. To save your changes:
 1. Choose Save G-Code As from the editor's File menu.
 2. Choose the drive you want from the Drives pull-down menu.
 3. Click on the folder you want in the Folder box.
 4. Type "TUTOR2.FGC" in the File name box.
 5. Choose OK.

5. To close the editor, choose Exit.

Going Online

Now you're ready to have FlashCut establish communications with the Signal Generator. This will put FlashCut into Online mode. Note that when the Signal Generator is online, all moves will be performed by the machine tool. If you're running the Demo version of FlashCut or do not have the means to go online at this time, ignore this section and continue with "Using the Jog Controls" below.

1. Make sure that Signal Generator is securely connected via serial cable to the correct serial port, as described in the Initial Setup section of this manual.
2. Turn on the Signal Generator.
3. Choose Online from the Controller menu.



4. After a short wait, FlashCut will display the Safety First message. Please read the message carefully. If you agree with the safety guidelines, click Agree.

Using the Jog Controls

The jog controls let you manually position the tool anywhere within the machine tool envelope.

1. Choose the Jog button in the Control Selection Box. FlashCut displays the Jog Control Panel.
2. Select the Slow radio button. The machine tool will move at the Slow Jog Rate defined in the Feedrate/Ramping Setup dialog box. If you are offline and/or running the Demo software, the jog rate on the screen will be very fast (determined by the speed of your computer).
3. Check the machine tool to make sure there's enough room to move the tool in the Z+ direction. If there is enough room, click down, hold and then release the Z+ Axis Jog button. Notice how the tool moved up and the Z Program and Relative coordinates changed. Also notice how the machine moved until you released the button. If the tool went down instead of up, change the Motor Polarity for the Z axis in the Motor Signal Setup dialog box. Note that if Machine coordinates are properly set, FlashCut will not let you move beyond the machine tool envelope. Also note that you can jog the machine using the keyboard. The controls are mapped as follows:

X+	Ctrl + Right Arrow Key
X-	Ctrl + Left Arrow Key
Y+	Ctrl + Up Arrow Key
Y-	Ctrl + Down Arrow Key
Z+	Ctrl + Page Up Key
Z-	Ctrl + Page Down Key
A+	Ctrl + Plus (+) Key
A-	Ctrl + Minus (-) Key

4. Try jogging in all directions on all axes, making sure you have enough room in the direction of travel before you choose each Axis Jog button.
5. Now select the Fast radio button and repeat the same exercise. The tool will move at the Fast Jog Rate defined in the Feedrate/Ramping Setup dialog box.
6. Position the tool so there is at least 1" of room in the positive Z direction.
7. Now let's set the current position as Program coordinates X=0, Y=0, Z=1. First, choose the Set button next to the Program label in the Tool Position Box. Then choose the Zero X and Zero Y buttons. Finally, enter 1.0000 in the Z text box. Choose OK to exit the dialog.
8. Now let's use the Axis Jog buttons to move the tool up exactly 1.0000". It's convenient to use three jog rates to do this: Fast, Slow and Single Step.
9. While the jog rate is still set to Fast, move the tool up until the Z axis Program coordinate is close to 2".
10. Now change the jog rate to Slow and do the same to get even closer to 2".
11. Finally, change the jog rate to Single Step. FlashCut will move the tool exactly one step each time you choose one of the Axis Jog buttons. Repeatedly choose either the +Z or -Z Axis Jog button until the tool is exactly at Program coordinate 2.0000 on the Z axis. Depending on the tool positioning resolution of your machine tool, you might not be able to reach 2.0000 exactly.
12. Now use the same process to move the tool exactly 0.5000 inches in the +X direction and 0.2500 inches in the +Y direction. The Program coordinates should now read:

X: 0.5000

Y: 0.2500

Z: 2.0000

Setting Machine Zero

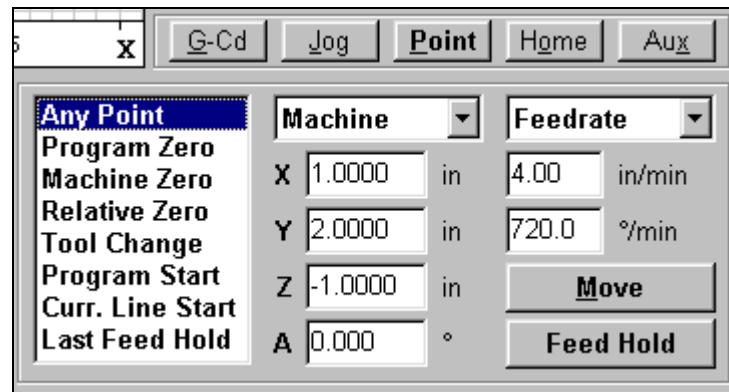
You can set Machine zero two ways: by jogging the tool to the corner of the machine tool envelope, or using the Seek Home button on the Home Control Panel. In this tutorial we will jog to a point and define it as Machine Zero.

1. Choose the Clear button next to the Machine coordinate label in the Tool Position Box. This will clear the Machine coordinates and remove the light blue machine tool envelope boundary.
2. Choose the Jog button in the Control Selection Box.
3. Jog the tool to 1/10" from the top of the Z axis.
4. Jog the table in the X- direction to about 1/10" from the end of travel.
5. Jog the table in the Y- direction to about 1/10" from the end of travel.
6. Choose the Zero (0) button to the right of the Machine coordinate label. Then choose OK in the confirmation dialog box that follows.

Using the Point Move

The Point Control Panel provides an easier way to move the tool to an exact position.

1. Choose the Point button in the Control Selection Box.
2. Select Any Point in the Point list.
3. Select Machine from the coordinate system pull-down menu.
4. Fill in the X, Y and Z text boxes to read 1, 2, and -1.
5. Enter "4" in the Linear Feedrate text box. Now everything is ready to move to Machine coordinates 1, 2, -1 at a feedrate of 4 inches/minute.



6. Choose the Move button. Notice how the machine first performed 2-Axis linear interpolation for the X and Y axis, then moved the Z axis down. This sequence of movement, unique to the Point mode and G28/G29 commands, helps to avoid tool crashes.
7. Select Machine Zero in the Point list.
8. Select Rapid from the Rate Mode pull-down menu. Everything is set to move the tool to Machine coordinate 0, 0, 0 as quickly as possible.

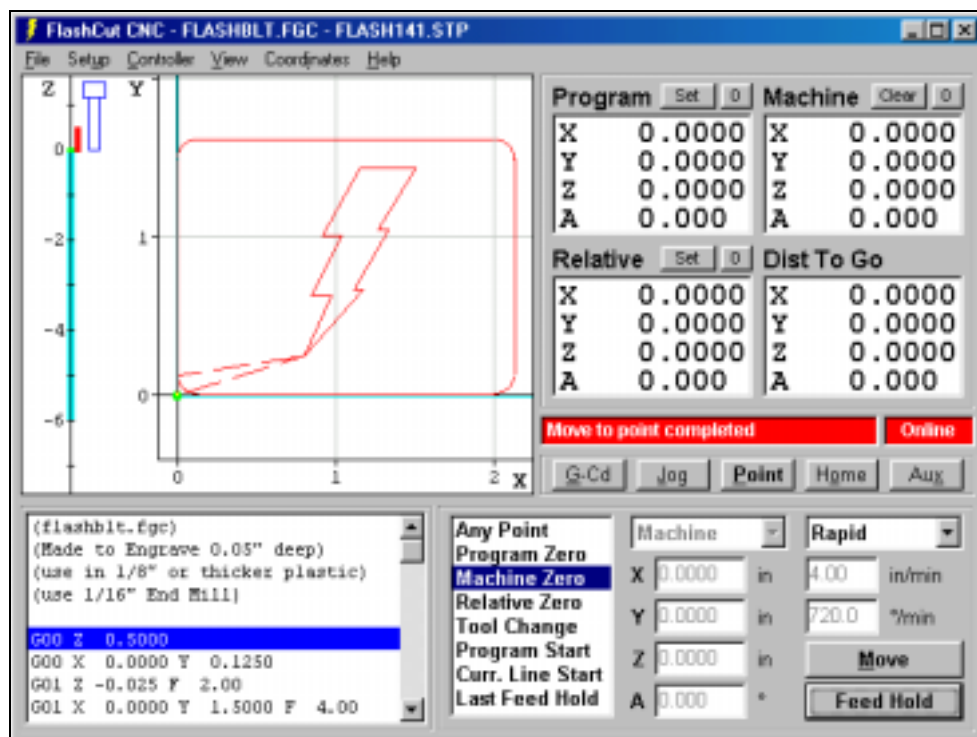


9. Choose the Move button. Notice how the machine first moved the Z axis up and then performed 2-axis linear interpolation on the X and Y axes.

Setting Program Zero on the Machine Tool

Program Zero is the origin to which all Program coordinates in the G-Code file are referenced. Before we cut a part, Program Zero must be set correctly relative to the workpiece. For this tutorial we will cut a file called FLASHBLT.FGC.

1. Choose Open G-Code from the File menu and select FLASHBLT.FGC, then choose OK. The screen appears like this:



2. Make sure there is enough room on all axes of the machine to run this G-Code file from the Program Zero point. By observing the toolpath displayed in the Toolpath View Port, you'll see the FLASHBLT.FGC program needs +2.125 inches on the X axis, +1.625 inches on the Y axis, +0.5 and -0.05 inches on the Z axis.

3. Fixture a sheet of 1/8" or thicker plastic or aluminum at least 2" wide and 4" long onto the XY table of the machine tool. Be sure that all clamps and fixtures are well out of the way of the tool during all parts of the program. This program works best with a 1/16" endmill.
4. Jog the tool to the -X, -Y corner of the workpiece using the Jog Control Panel. Then carefully jog the tool down in the Z axis until it just barely touches the top of the workpiece.
5. To define this point as Program Zero, choose the Zero (0) button next to the Program label, then choose OK in the dialog that immediately follows.
6. Using either the Jog or Point Control Panel, raise the Z axis exactly 1.0" and define this point as Program Zero. Note that this point is exactly 1" above where it needs to be to actually cut the workpiece. The machine is now ready to do a dry run without cutting the workpiece.

Testing the Program on the Machine Tool

It is always a good idea to do a dry run of the G-Code file both in offline and online modes before cutting a part. This lets you check for G-Code programming errors, spot interferences with fixtures, and so on.

Choose the G-Cd button in the Control Selection Box.

1. Choose the Reset button to make sure the G-Code program is reset. Choose OK in the dialog that follows.
2. Take the Signal Generator offline by choosing Offline from the Controller menu.
3. Go into step mode by choosing the Step radio button.
4. Repeatedly choose the Start button, watching the screen to make sure the tool behaves properly.
5. Once you are satisfied the program will behave properly, re-establish communications with the Signal Generator by choosing Online from the Controller menu. A dialog will ask you if you want to revert to the coordinates used before going offline. Choose Yes.
6. Choose the Step radio button.
7. Choose the Start button. The machine should move the X and Y axes, then stop.
8. Step through the entire program by choosing the Start button for every line of G-Code. If at any time you need to stop the tool, choose the Feed Hold button or hit any key on your keyboard. If you stop the tool in the middle of the program, you can start exactly where you left off by choosing the Start button. You may want to try this for practice.

Cutting the Part

Assuming everything was fine in the previous step, we're ready to cut a part.

1. Check to make sure the Program coordinates are at 0,0,0. If not, go through the "Setting Program Zero on the Machine Tool" section above.
2. Go into jog mode and **carefully** move the tool down in the Z- direction to the part surface. This should be at the program coordinate 0,0,-1.
3. Set this point as Program Zero by choosing the Zero (0) button next to the Program label and choosing Yes in the dialog that follows.
4. Now jog the tool 0.5" **UP** (in the Z+ direction). This places the tool in the correct starting position to begin cutting the workpiece.
5. Choose the G-Cd button in the Control Selection Box.
6. Choose the Continuous radio button.
7. Turn the machine tool spindle on and make sure everything is ready on the machine tool.
8. Choose the Start button and the machine tool will begin to cut out your first part. **Always be on alert to choose the Feed Hold button or hit any key in case of emergency.**
9. Congratulations! You've successfully cut your first part using FlashCut.

Exiting the Program

To exit FlashCut CNC, choose Exit from the File menu. This terminates communications with the Signal Generator and returns you back to Windows.

Turning off the Electronics

Always turn off the Signal Generator and Motor Driver when they are not in use.

Section 6. Glossary

- Backlash** - The amount of motor movement that occurs without table movement when changing directions on an axis. A typical source for backlash is play between the screw and nut in the drive system.
- Baud Rate** - The speed at which FlashCut communicates across the serial port with the Signal Generator. It's measured in bits per second and is typically set at 38,400.
- Buffer Time** - The Buffer Time is used to prevent system events from affecting motor movement. The larger the Buffer Time, the less effect system events have on motor movement. The smaller the Buffer Time, the more responsive the machine tool is to mouse clicks.
- CNC Setup Parameters** - FlashCut CNC settings that define a machine tool and its operating limits. All setup parameters are found in dialog boxes available from the Setup menu. The coordinate display mode and the scale to fit setting are also saved in the setup file.
- Command Buttons** - Buttons that perform a task when chosen.
- Continuous Contouring** - A high performance operating mode that lets FlashCut run G-code files smoother and faster. Smooth transitions from one G-code line to the next allow high speed machining of complex 2D profiles and 3D surfaces. Also called "Look-Ahead" and "Constant Contouring."
- DXF** - Document eXchange Format. Defined by AutoDesk Inc. as a way to exchange design data between CAD, CAM and CAE programs.
- Feedrate** - The linear speed of the cutting tool relative to the workpiece. Defined in G-Code by the F parameter in inches/minute, millimeters/minute or degrees/minute.
- Full-Step** - Step mode where one step from the Signal Generator corresponds to one full step of a stepper motor.
- G-Code** - Standard programming language used to control CNC machines.
- G-Code Program** - Program used to control the movement of the machine tool. G-Code is an industry-standard machine tool programming language.
- Gear Ratio** - The ratio of motor revolutions to drive screw revolutions due to gears, pulleys or other devices.
- Half-Step** - Step mode where two steps from the Signal Generator correspond to one full step of a stepper motor.
- Included File** - G-Code file containing subroutines that can be called from a main G-Code file. The included file must be listed at the top of the main G-Code file.
- Jog** - Method of manually controlling each axis of the machine tool.
- Limit Switches** - Switches placed at the travel limits of each axis. When the machine tool travels too far in either direction of any axis, a limit switch is tripped. The control system then stops the machine to prevent damage.
- Machine Coordinates** - The XYZA position of the tool relative to Machine Zero.
- Machine Origin** - Same as Machine Zero.

Machine Tool Envelope - The three dimensional box (plus optional 4th axis travel) defined by the maximum travel in the X, Y, Z and A axes. Once the machine tool envelope is defined, the tool cannot move beyond it. You can disable the machine tool envelope by choosing the Clear button.

Machine Zero - The origin (X, Y, Z, A = 0, 0, 0, 0) of useful space within the machine tool envelope. Can either be defined manually or by using home switches.

Maximum Feedrate - The maximum rate at which a motor can reliably start and stop with ramping.

Maximum Unramped Feedrate - The maximum rate at which a motor can reliably start and stop without ramping.

Modal - A type of command that stays in effect while FlashCut processes subsequent G-Code commands.

Motor Driver - The electronic box that converts step and direction signals from the Signal Generator into a sequence of amplified signals to drive a motor.

Motor Polarity - The association between the actual direction an axis moves and the direction FlashCut intends it to move. If they are different, this setting should be changed from positive to negative or vice-versa.

Motor Resolution - The number of full motor steps for one revolution of the motor. For example, a 1.8° Stepper motor has 200 full steps per revolution, a 0.9° Stepper Motor has 400 full steps per revolution, and so on.

Motor Step - The amount of motor movement associated with one electrical pulse to the stepper motor driver.

Offline - Mode in which FlashCut does not communicate with the Signal Generator. FlashCut displays all G-Code, Jog, and Point moves, but the machine tool does not move.

Online - Mode in which FlashCut communicates with the Signal Generator. In this mode, all G-Code, Jog and Point moves are executed by the machine tool.

Open Loop - A type of control system in which an actuator moves without sending motion information back to the controller. Most stepper motor systems are open loop due to their high reliability in performing step commands when used within their torque limits.

Program Coordinates - The XYZA position of the machine tool relative to Program Zero.

Program Listing Box - The area of the main screen that displays the G-Code program currently loaded into FlashCut.

Program Zero - The zero point, or origin, to which all absolute coordinates in the G-Code file are referenced. It is depicted as a green dot in the Toolpath View Port.

Pull-Down Menu - A standard Windows control that lets the user select a single item from a list.

Radio Buttons - Buttons representing a group of options requiring a single selection.

Ramping - Smooth acceleration that allows a machine tool to reach high feedrates. Ramping rates are measured in full steps/sec/sec.

Relative Coordinates - The XYZA position of the machine tool relative to the point at which the Relative Coordinates were zeroed. The relative coordinate system is general purpose and may be used for anything you choose (as long as it's legal).

Resonant Speeds - Rotational speeds at which a stepper motor vibrates excessively. Often the motor will stall if run at these speeds. Resonant speeds depend on the size of the motor, the amount of loading, and the power of the motor driver. Typically, increasing the load and reducing motor driver current will reduce resonance.

Screw Thread - The number of turns per unit length of travel of the helical drive screw for each axis.

Serial Port - A communications port on both the PC and the Signal Generator, used to exchange commands and other information.

Setup File - A file containing the CNC Setup Parameters for a machine tool. These files have a “.STP” extension by default.

Signal Generator - The electronic box that converts computer commands into step and direction signals suitable for a stepper or digital servo motor driver. It also interprets input line signals and produces output signals to control various peripheral devices.

Step Mode - The number of micro steps between each full motor step.

Stepper Motor - A motor that moves a precise amount when given an electrical pulse. Stepper motors typically have 200 full steps per revolution, or 1.8° per full step. Other popular stepper motors have 0.9° and 7.5° per step.

Text Boxes - Areas in which you type either a name or a value.

Tool Position Box - The area of the main screen that shows the current coordinates of the tool on the machine tool. You can display one or all of the four coordinate systems in this box.

Tool Positioning Resolution - The amount of machine tool movement that results from one step pulse produced by the Signal Generator. For each axis, Tool Positioning Resolution (TPR) depends on 4 factors: Step Mode (SM), Gear Ratio (GR), Motor Resolution (MR), and Screw Thread (ST), where

$$\text{TPR} = 1 / (\text{SM} * \text{GR} * \text{MR} * \text{ST})$$

Toolpath - The path that a machine tool follows as FlashCut executes a G-Code program.

Toolpath View Port - The area of the main screen that graphically displays the toolpath in real time.