# 5. Tutorial

Starting FlashCut CNC

To start FlashCut CNC, click on the Start button, select Programs, select FlashCut CNC 4, then select the FlashCut CNC 4 icon.

A dialog will appear asking you if you want to start with the Signal Generator connected. Choose the Don't Connect button. If you're running the Demo version, choose Continue.

Configuring FlashCut CNC

1. Choose New Setup from the File menu.
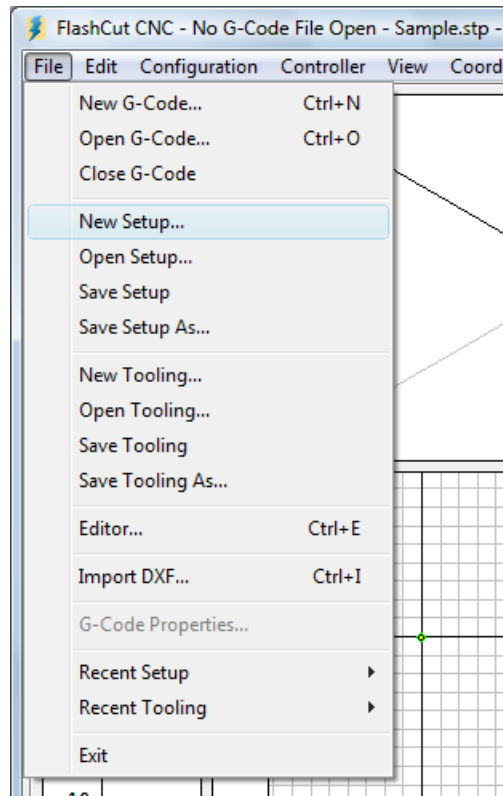
2. The configuration wizard will appear. Answer the questions in the wizard to create a default setup file for your machine. If you're using the Demo version of the software, you don't need to do anything else here. If you're using the live version of the software and running your machine tool, you'll need to finish configuring the setup file for your machine. Please refer to Initial Setup section of this manual for more details.

3. Choose DRO:All Coordinates from the View menu to make sure all four coordinate systems are displayed.

4. Choose Viewports from the Configuration menu. Set the View On checkboxes so only the following viewports are displayed: Z, XY, and XYZ.

Loading a G-Code File

1. Choose Open G-Code from the File menu.
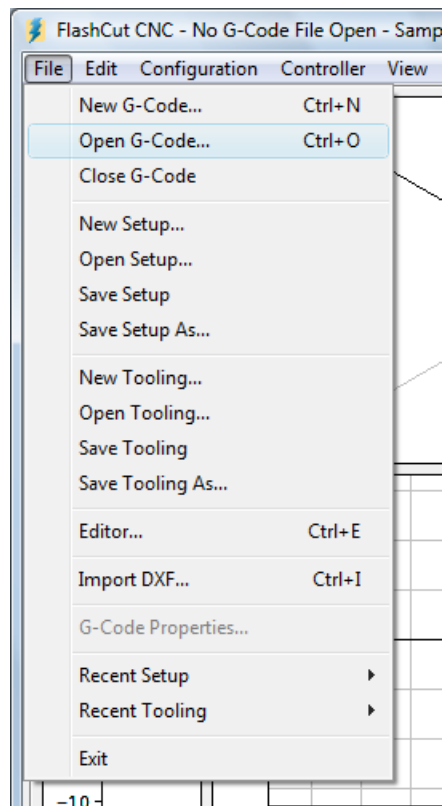
2. Double-click on the file TUTOR.FGC.
   FlashCut will load TUTOR.FGC and the screen should now look like this:
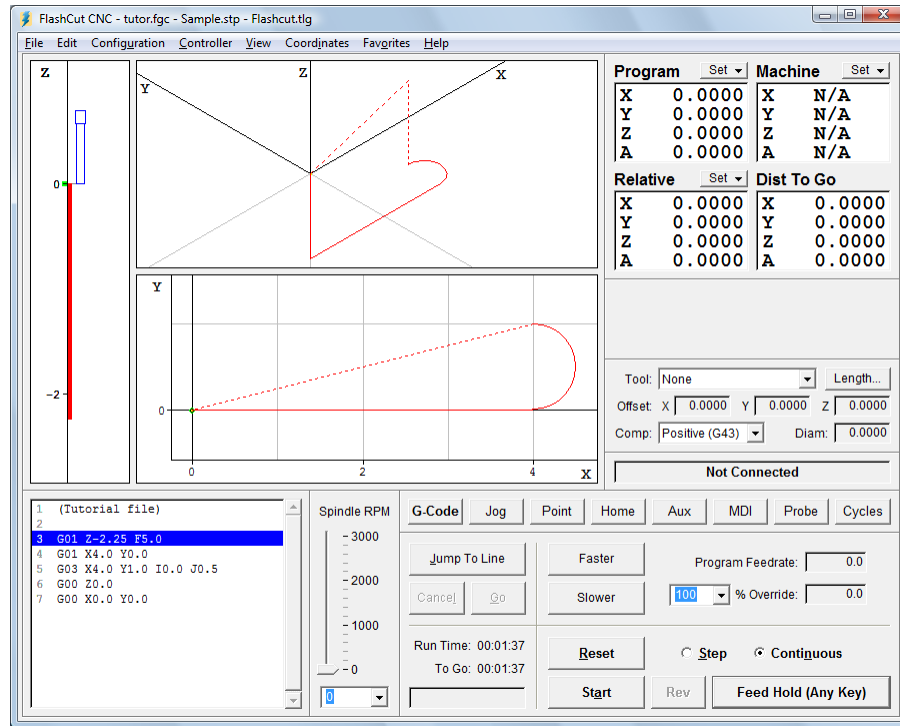
Notice how the G-Code listing appeared in the Program Listing Box and a red outline of the toolpath appeared in the Viewport Box.

Viewing the Toolpath

Before examining the toolpath display let's zero the machine coordinates. To do this, choose Zero All from the Set button to the right of the Machine label in the DRO 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 Viewport Box. Because Auto Zoom Toolpath Extents is checked in the View menu, you'll only see a portion of the tool envelope boundary, as shown below.
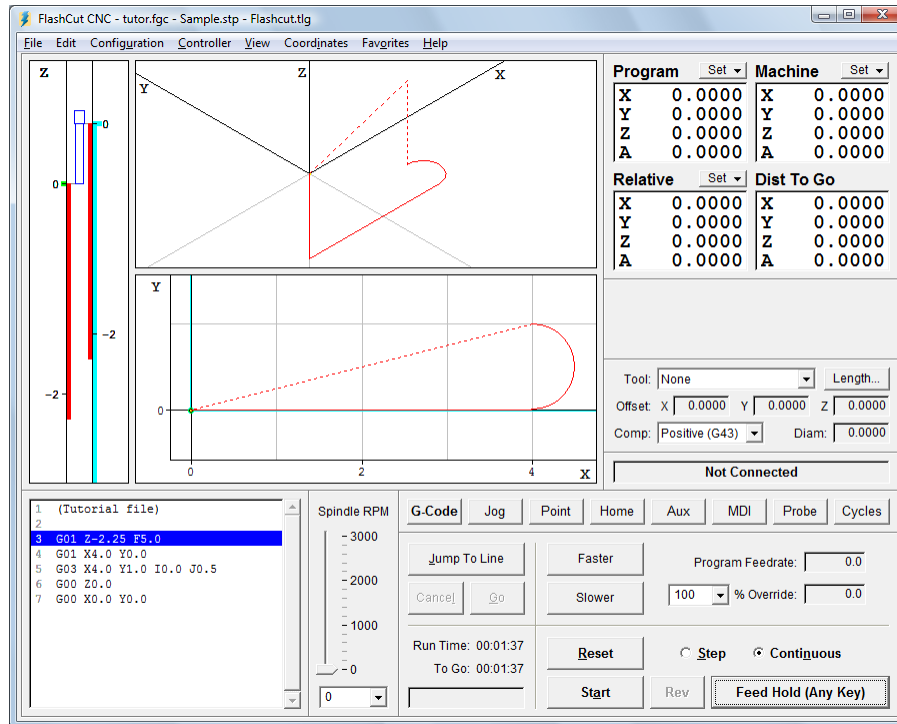
To see the entire machine tool envelope, choose Auto Zoom Machine Extents from the View menu. The screen will then appear like this:
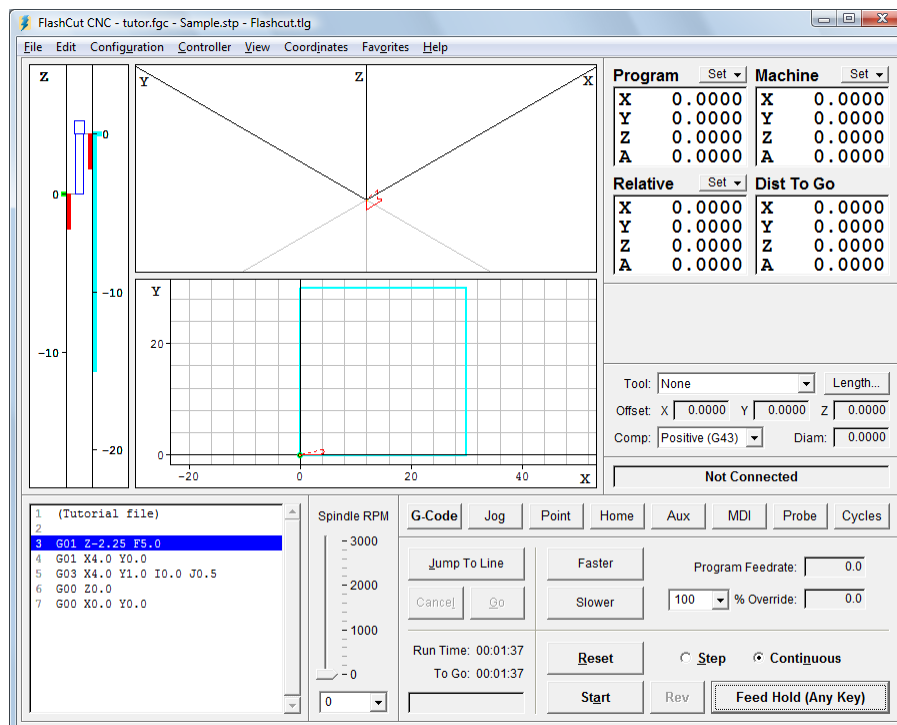
Choose Auto Zoom Toolpath Extents from the View menu again, to see the largest possible image of the toolpath.

Now let's get familiar with the Viewport Box. 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.

Thick 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.

Tool Icon – Represents the current Z axis position relative to the workpiece (bottom of icon) and relative to the machine (near top 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. On the Controller menu, make sure the Simulate When Not Connected command is checked.

2. Choose Zero All from the Set button next to the Program label, and then choose OK in the confirmation dialog box that follows. FlashCut sets all program coordinates to zero. This simulates the tool being in the correct position before the G-Code program begins.

3. Choose the G-Code button in the Control Selection Box to make sure the G-Code Control Panel is displayed.

4. Select the Step radio button so FlashCut will execute the G-Code program one line at a time.

5. Choose the Start button and watch the tool icon move down the Z axis viewport. Notice that FlashCut has highlighted the next line in the Program Listing Box, indicating it has fully executed the first line.

6. 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.

7. Select the Continuous radio button, and then choose the Start button again. The yellow dot moves along the toolpath to completion, leaving a blue trace behind.

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.
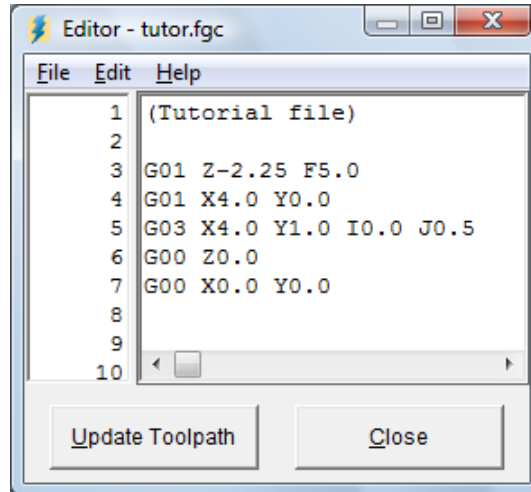
2. First, let's reduce the cutting depth from 2.25" to 0.25". With your mouse or your arrow keys, move the cursor to the first line of G-Code that reads: G01 Z-2.25 F5.0. Change it to read: G01 Z-0.25 F5.0. Then choose Update Toolpath at the bottom of the editor dialog box. Notice how the red line in the Z viewport, 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 X4.0 Y1.0 I0.0 J0.5. Change it to read: G03 X4.0 Y2.0 I0.0 J1.0. Choose Update Toolpath to see the effect of the change.

4. To save your changes:

    *   Choose Save G-Code As from the editor's File menu.

    *   Type "TUTOR2.FGC" in the File name box.

    *   Choose OK.

5. To close the editor, choose Close.

Connecting with the Signal Generator

Now you're ready to have FlashCut establish communications with the Signal Generator. Note that when the Signal Generator is connected, 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 connect to the Signal Generator at this time, ignore this section and continue with "Using the Jog Controls" below.

1. Make sure that Signal Generator is securely connected via USB cable to your PC, and you've completed the USB driver installation.

2. Turn on the Signal Generator.

3. Choose Connect from the Controller menu.
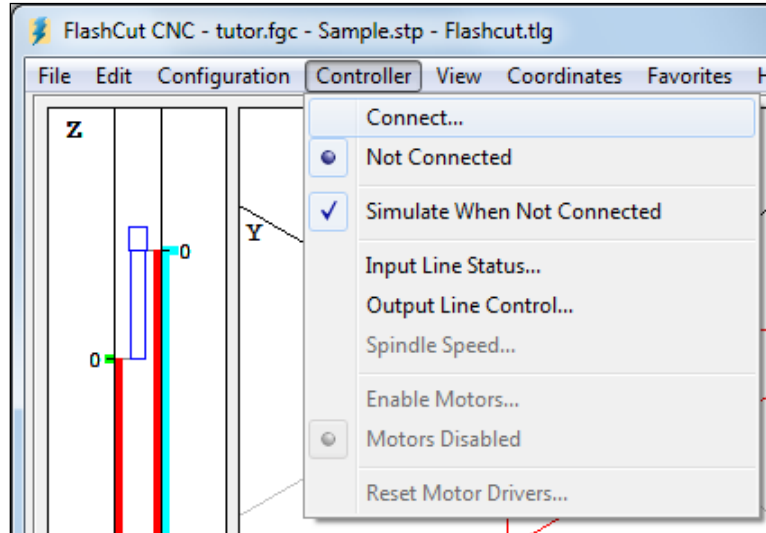
4. FlashCut will display the Safety First message. Please read the message carefully. If you agree with the safety guidelines, click Agree.

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 and Contin (Continuous) radio buttons. The machine tool will move continuously at the slow jog rate (defined on the Jogging panel of the Configuration dialog box.)

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+ 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 Direction for the Z axis on the Motor Signals panel of the Configuration dialog box. Note that when machine coordinates are being used, 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 on the Jogging panel of the Configuration 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 Enter from the Set button next to the Program label in the DRO 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 jog buttons to move the tool up exactly 1.0000". It's convenient to use three jog modes to do this: Fast Continuous, Slow Continuous and Slow 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 mode to Step. FlashCut will move the tool exactly one step each time you choose one of the 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

In general you can set Machine Zero two ways: manually by jogging the tool to the corner of the machine tool envelope and choosing Zero All from the machine coordinates Set button, or by using the Seek Home button on the Home Control Panel. In this tutorial we will jog to a point and manually define it as Machine Zero. (To use the Seek Home button, your machine must have homing switches installed.)

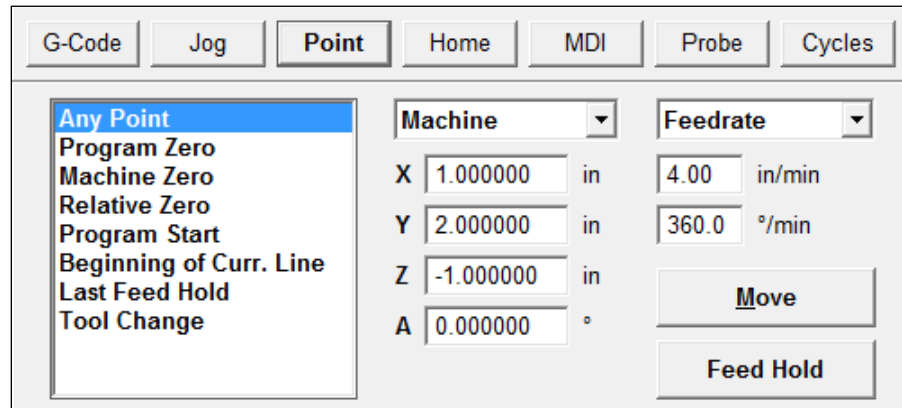Follow these steps to set Machine Zero manually.

1. Choose Clear from the Set button next to the machine coordinate label in the DRO 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 Zero All from the Set button to the right of the machine coordinate label. Then choose OK in the confirmation dialog box that follows.
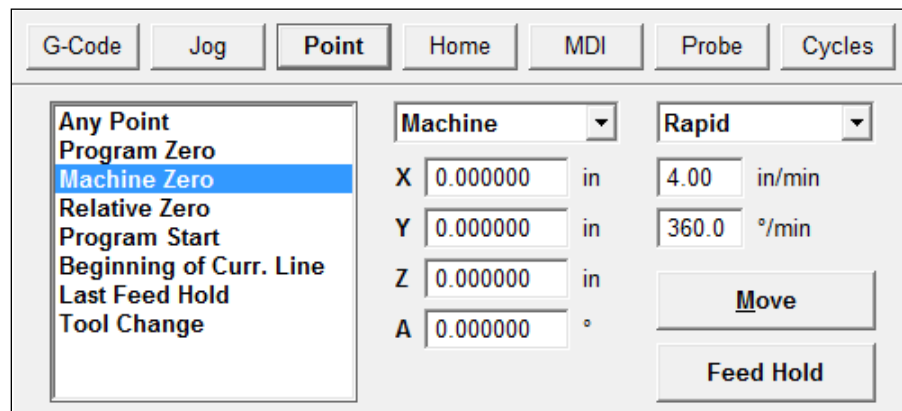
Using the Point Move

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.

G-Code	Jog	**Point**	Home	MDI	Probe	Cycles

Any Point	Machine ▼	Feedrate ▼
Program Zero		
Machine Zero	X 1.000000 in	4.00 in/min
Relative Zero		
Program Start	Y 2.000000 in	360.0 °/min
Beginning of Curr. Line		
Last Feed Hold	Z -1.000000 in	**Move**
Tool Change	A 0.000000 °	
		Feed Hold

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/G30 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.

G-Code	Jog	**Point**	Home	MDI	Probe	Cycles

Any Point	Machine ▼	Rapid ▼
Program Zero		
Machine Zero	X 0.000000 in	4.00 in/min
Relative Zero		
Program Start	Y 0.000000 in	360.0 °/min
Beginning of Curr. Line		
Last Feed Hold	Z 0.000000 in	**Move**
Tool Change	A 0.000000 °	
		Feed Hold

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 FLASHBOLT.FGC.

1. Choose Open G-Code from the File menu and select FLASHBOLT.FGC, and then choose OK. The screen appears like this:
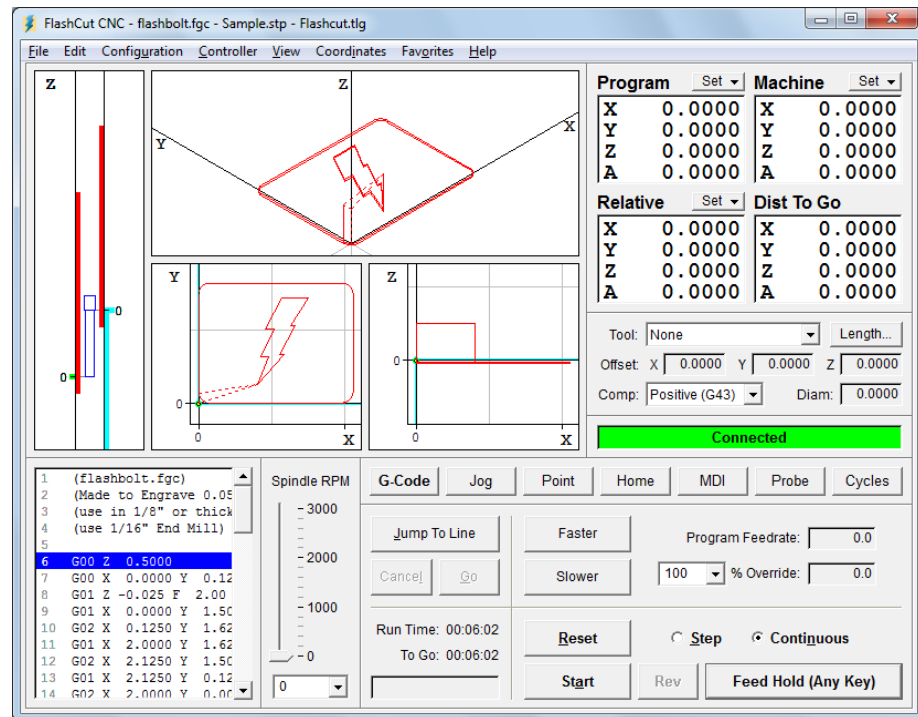
2. Make sure there is enough room on all axes of the machine to run this G-Code file from the Program Zero point. By observing the toolpath displayed in the Viewport Box, you'll see the FLASHBOLT.FGC program needs +2.125 inches on the X axis, +1.625 inches on the Y axis, +0.5 and -0.05 inches on the 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" end mill.

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 Zero All from the Set 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 connected and not connected to the Signal Generator, before cutting a part. This lets you check for G-Code programming errors, spot interferences with fixtures, and so on.

1. Choose the G-Code button in the Control Selection Box.

2. Choose the Reset button to make sure the G-Code program is reset. Choose OK in the dialog that follows.

3. Disconnect from the Signal Generator by choosing Disconnect from the Controller menu.

4. Go into step mode by choosing the Step radio button.

5. Repeatedly choose the Start button, watching the screen to make sure the tool behaves properly.

6. Once you are satisfied the program will behave properly, re-establish communications with the Signal Generator by choosing Connect from the Controller menu. A dialog will ask you if you want to revert to the coordinates used before disconnecting. Choose Yes.

7. Choose the Step radio button.

8. Choose the Start button. The machine should move the X and Y axes, and then stop.

9. 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 Zero All from the Set 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-Code 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, 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.