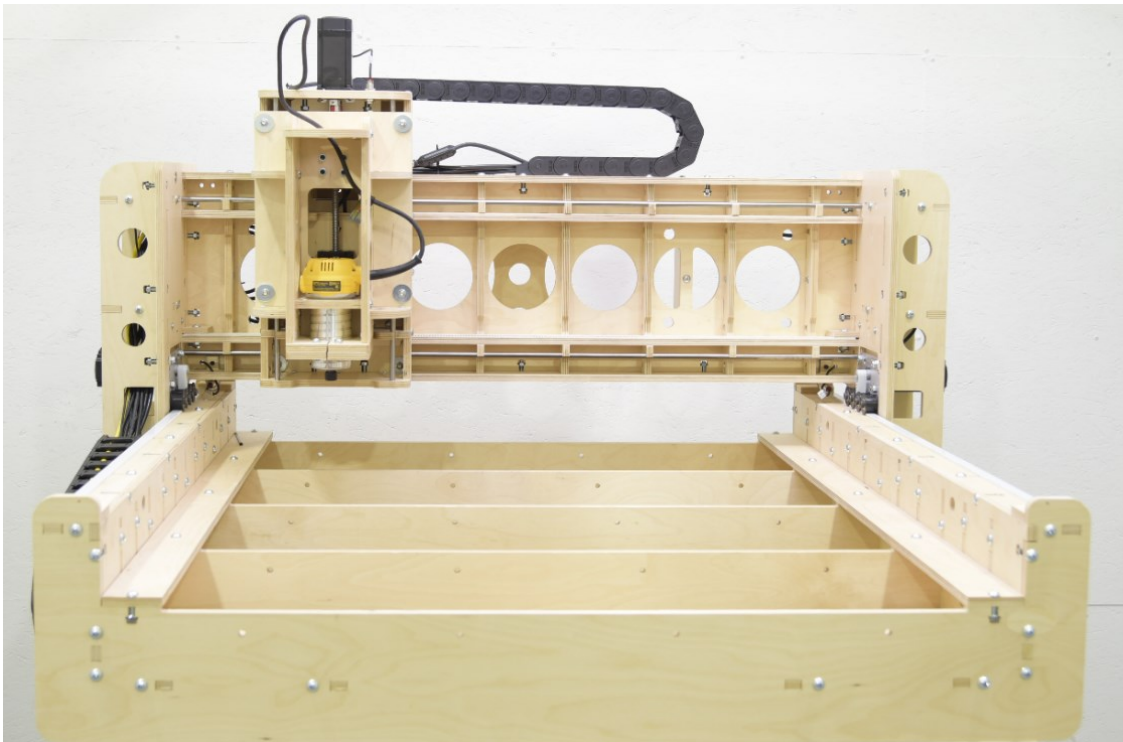# BobsCNC
Unleash Your Creativity

KL7 Series

Software and First Project

MANUAL 8 of 8

Version 1.6

Table of Contents

⚠	**DANGER** Indicates a serious risk of bodily harm, possible injury and death. This warning box is to be taken seriously. Any work must be carried out with extreme caution.

⚠	**CAUTION** Indicates a possible risk of injury that can result from failure to follow this warning.

⚠	**WARNING** Indicates the possible damage to the machine, its components, or the work piece that can result from the failure to follow this warning.

💡 HINT	Hints will provide needed information, shortcuts, and insights that will make assembly and machine operation easier and safer.

Please review each assembly manual before beginning to assemble the KL7 SERIES CNC Router.

Choosing the Software

To complete a project using your CNC router you need 3 types of software.

1. **CAD**—**C**omputer **A**ided **D**esign software is the software you use to create designs. CAD can be as simple drawing program or a 3D model maker.

2. **CAM**—**C**omputer **A**ided **M**achining software takes the CAD file or work and aids in creating tool paths. A tool path simply gives a command of where, how fast, and how deep the router bit should travel. The completed tool path file is typically referred to as a gcode file.

3. **Gcode Sender**—Gcode sender software does exactly what its name implies. It takes the gcode file you have created and sends it to the CNC router.

NOTE: Some software packages combine CAD/CAM and even a Gcode sender into one package.

CAD and CAM software for 3 axes CNC routers can be separated into 2 groups by the type of projects they can help create.

1. 2.5D— projects include engravings and cutouts. The Z is fixed during the X and Y movements.
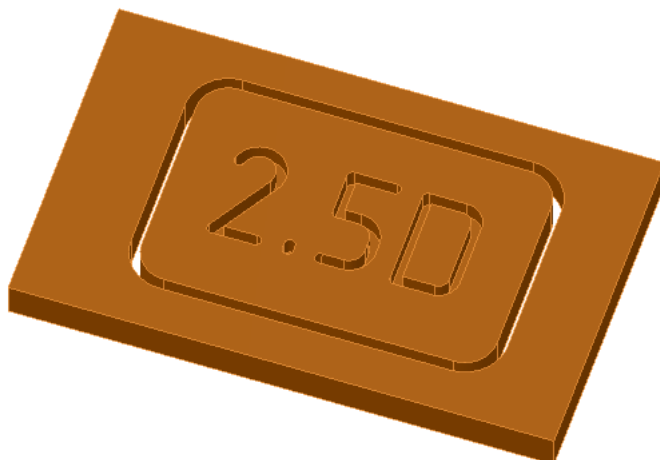
2. 3D— projects include contour cuts where the CNC's Z axis is moving with the X and Y axes to create the 3D surfaces. Below is a simple 3D model.

NOTE: There are CNC routers that also have 4th and 5th Axes that can cut more complex shapes. 4th and 5th axes CNC router software is needed to take advantage of the extra rotating axis.

The KL7 Series use an UNO microcontroller running *grbl1.1* to interpret the gcode files. You will need a gcode sender compatible with *grbl1.1*. We recommend the *UGS Platform* for sending gcode files to the CNC. This software is free and open source.

There are many CAM and CAM software choices. Some are free (Open Source). The cost of others depends on the features they offer.   Here's a list of some you may wish to consider.

> Vectric VCarve Desktop, Pro, and Aspire
> EstlCam
> Easel
> F-engrave
> Carbide Create
> MeshCAM
> Fusion 360
> dmap2gcode

Installing and Checking the Serial Driver

Step 1   Download and save the Arduino IDE software from:

https://www.arduino.cc/en/Main/Software

Click the Windows installer (or other OS)

Click the Contribute & Download or Just Download and save the file.

Step 2

Plug in the USB from the controller to the computer and browse to the **Download** folder and double click the Arduino Software installation file.

Note: The version may be different than the one shown.

arduino—1.8.8-windows.exe

NOTE: The Arduino IDE software enables users to program AVR microcontrollers. You DO NOT need this software. You only need the drivers to be installed.

If you accidently programmed the controller, it can be re-programmed so that the original firmware is installed. Please see the BobsCNC support page here:

https://support.bobscnc.com/hc/en-us/articles/360008797214-Controller-Firmware-Installation

Step 3   While the USB is still connected , open the Device Manager

Com port section.

NOTE: For a Windows 10 you can search for the Device Manager using the search textbox as shown.

NOTE: In the picture, the com port is COM3. Your com port may be different. Please note the number of the com port as you will need it to connect the controller to the CNC in a later step.
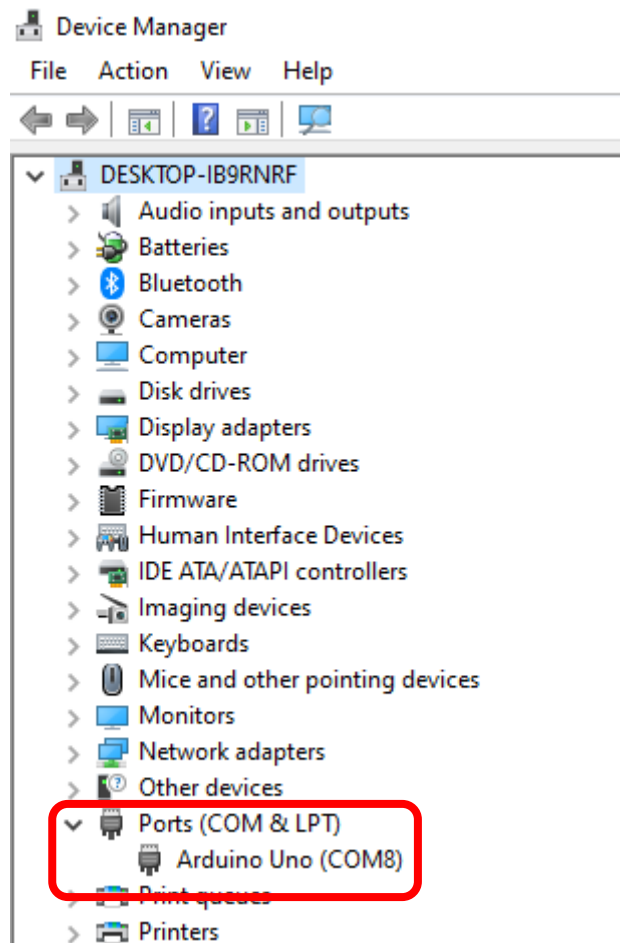
Installing the grbl firmware

Step 1   Download and save the BobsCNC Flash Zipped.exe from:

https://support.bobscnc.com/hc/en-us/articles/360008797214-Controller-Firmware-Installation

BobsCNC
Flash
Zipped.exe

Step 2 The BobsCNC Flash Zipped.exe is a self extracting zip file. Please double click on the file and save to the desired folder.
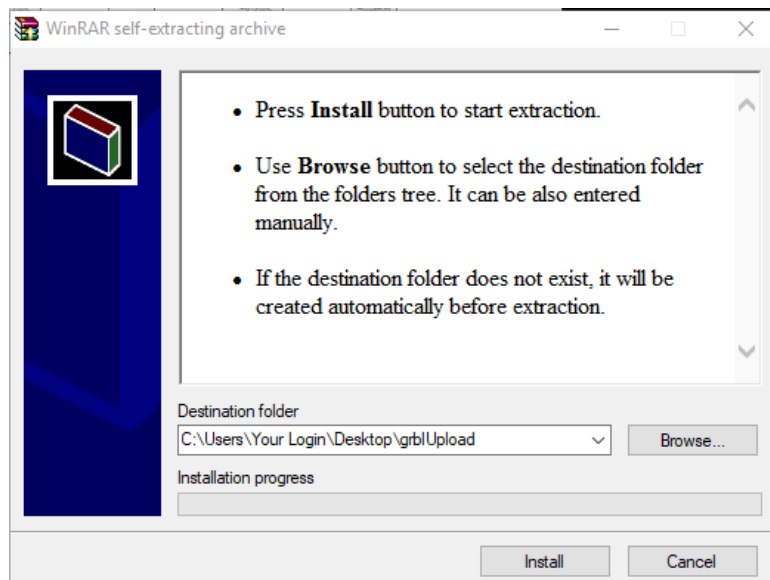
Step 3 Connect the CNC's USB to the computer and run the batch file for your CNC.

- The default KL733 CNC firmware will use the **KL733 bat** file.
- The default KL744 CNC firmware will use the **KL744 bat** file.

Step 4 When you run the batch file the CMD window will open and prompt

you for the com port. Please type the correct com port number and press the

```
C:\Windows\system32\cmd.exe                                              —    □    ×
This routine will clear the KL744 Series Controller, then re-flash the grbl1.1h firmware.
Please Enter the com port number that the KL744 is connected to:
```

HINT	Ensure that the CMD Window shows that the firmware is reading and writing. If it is not then, please note the error. Make sure the com port is correct. You may also need to remove and reinstall the USB cable and try again.

```
C:\Windows\system32\cmd.exe                                              —    □    ×
avrdude: Device signature = 0x1e950f
avrdude: NOTE: FLASH memory has been specified, an erase cycle will be performed
         To disable this feature, specify the -D option.
avrdude: erasing chip
avrdude: reading input file "EEPROM_CLEAR.hex"
avrdude: input file EEPROM_CLEAR.hex auto detected as Intel Hex
avrdude: writing flash (764 bytes):

Writing | ################################################## | 100% 0.12s

avrdude: 764 bytes of flash written
avrdude: verifying flash memory against EEPROM_CLEAR.hex:
avrdude: load data flash data from input file EEPROM_CLEAR.hex:
avrdude: input file EEPROM_CLEAR.hex auto detected as Intel Hex
avrdude: input file EEPROM_CLEAR.hex contains 764 bytes
avrdude: reading on-chip flash data:

Reading | ################################################## | 100% 0.10s

avrdude: verifying ...
avrdude: 764 bytes of flash verified

avrdude: safemode: Fuses OK

avrdude done.  Thank you.

Waiting to give time for the Clear EEPROM program to run

Waiting for  5 seconds, press CTRL+C to quit ...
```
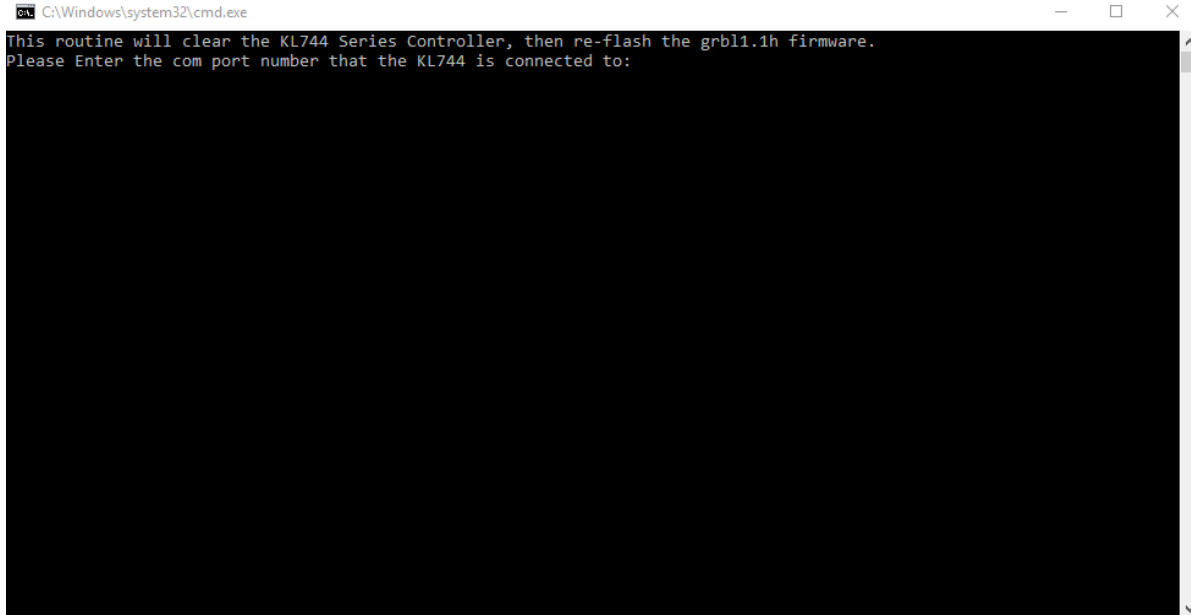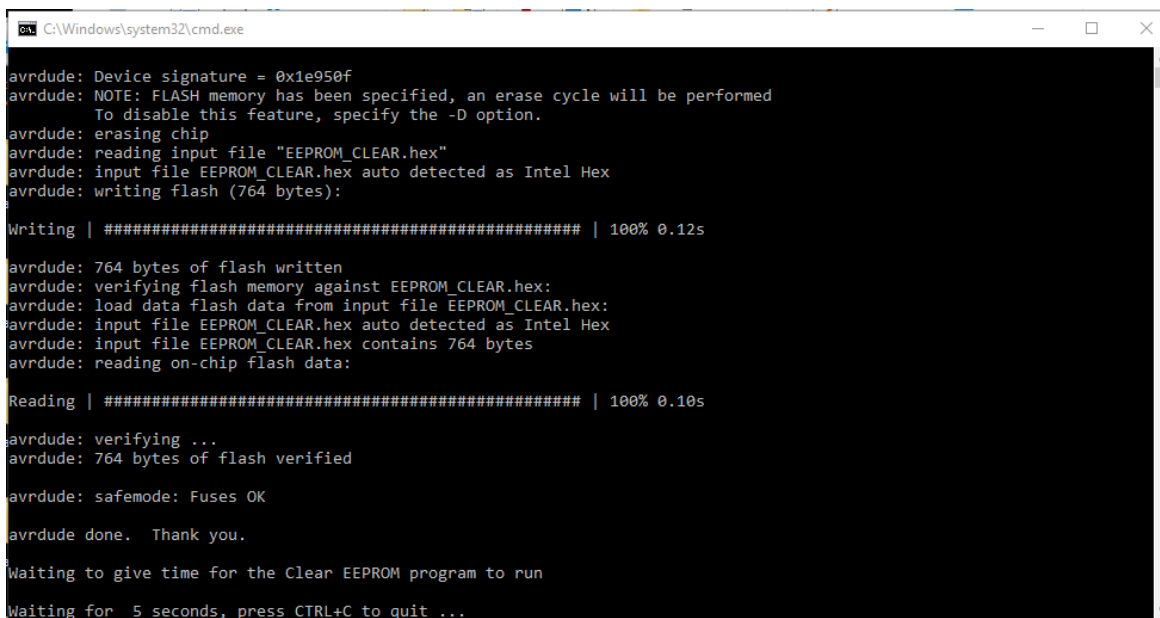
Once the batch file is complete you will be prompted to press an key to continue.

The firmware is loaded on the UNO.

Downloading UGS Platform and Required Java

Step 1   Download and save the Stable version UGS Platform software
as shown from:

https://winder.github.io/ugs_website/download/

Stable

These are considered stable releases. For even older releases please visit github

Version	Java Version	Release Date
2.0 Classic (Beta)	Java 8+	August 14, 2019
2.0 Platform (Beta)	Java 8+	August 14, 2019
1.0.9 Classic	Java 7+	November 11, 2015
1.0.8 Classic	Java 7+	February 25, 2015
1.0.7 Classic	Java 7+	March 30, 2014
1.0.6 Classic	Java 6+	March 15, 2013

Step 2   Download and save the Java 8+ software.

Stable

These are considered stable releases. For even older releases please visit github

Version	Java Version	Release Date
2.0 Classic (Beta)	Java 8+	August 14, 2019
2.0 Platform (Beta)	Java 8+	August 14, 2019
1.0.9 Classic	Java 7+	November 11, 2015
1.0.8 Classic	Java 7+	February 25, 2015
1.0.7 Classic	Java 7+	March 30, 2014
1.0.6 Classic	Java 6+	March 15, 2013

Step 3

Download and save the version of Java for your computer. Most newer computers will be the 64-bit as shown.

Select the file according to your operating system from the list below to get the latest Java for your computer.

> Remove Older Versions   > What is Java?

By downloading Java you acknowledge that you have read and accepted the terms of the Oracle Technology Network License Agreement for Oracle Java SE

Windows	ℹ Which should I choose?		
⬇	Windows Online filesize: 1.97 MB	Instructions	After installing Java, you may need to restart your browser in order to enable Java in your browser.
⬇	Windows Offline filesize: 64.93 MB	Instructions	
⬇	Windows Offline (64-bit) filesize: 72.8 MB	Instructions	

If you use 32-bit and 64-bit browsers interchangeably, you will need to install both 32-bit and 64-bit Java in order to have the Java plug-in for both browsers. » FAQ about 64-bit Java for Windows

Step 4

Browse to the **Download** folder and double click the Java Software installation file.

Note: The version may be different than the one shown.

jre-8u221-windows-x64.exe

Step 5

Browse to the **Download** folder and double click the UGS Platform Software zip file and unzip it to a known location.

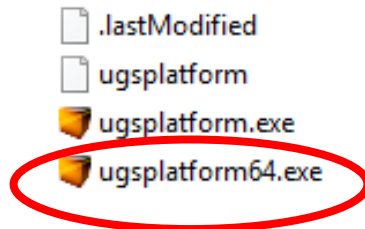Note: The version may be different than the one shown.
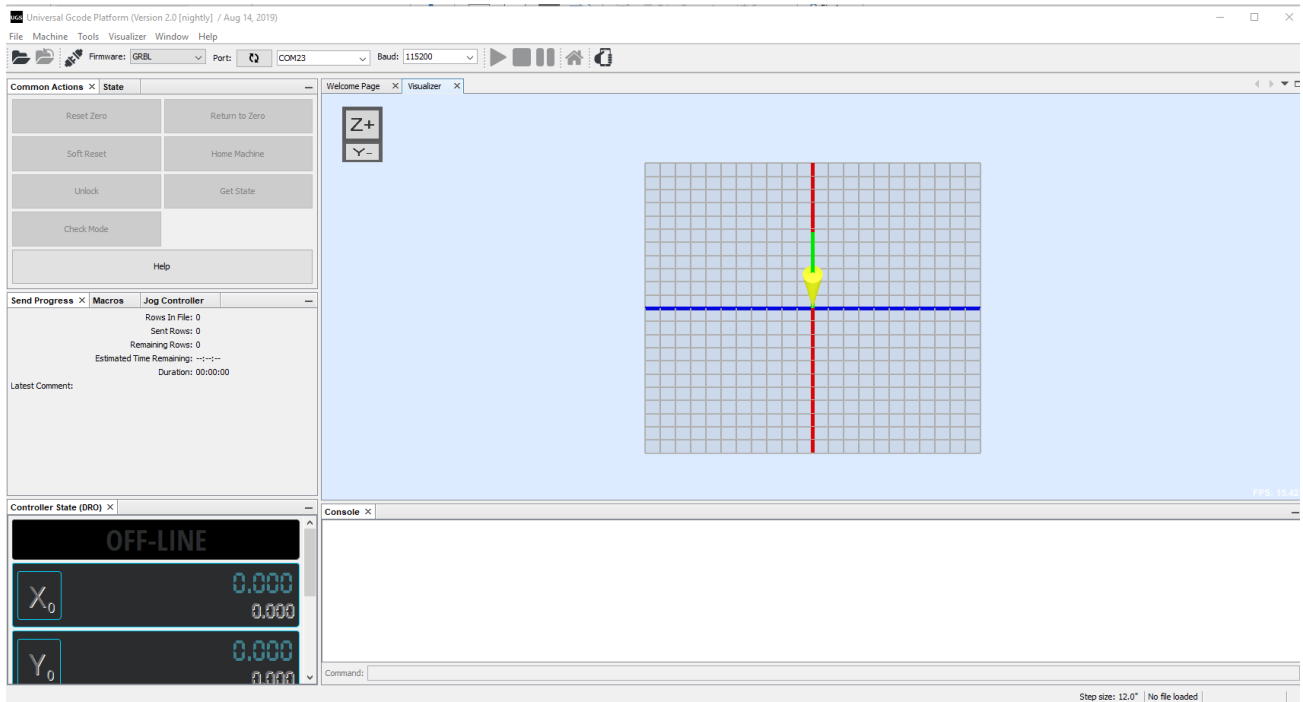
 ugsplatform-2.0.0-Aug.14.2019.zip

Step 6

Browse to the **Bin** folder in the unzipped **UGS Platform** folder and double click the **UGS platform** application file.

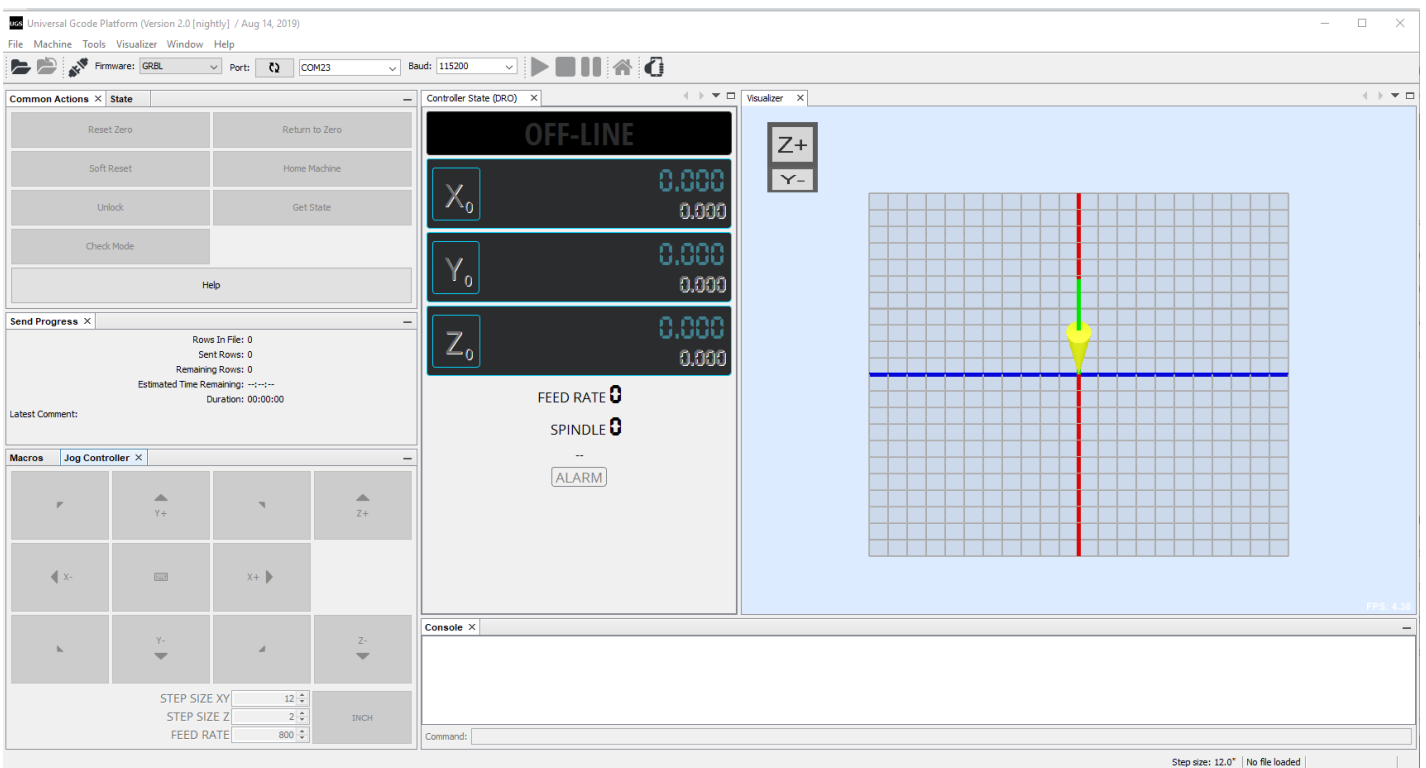Note: The if the 7usplatform64 does not work, try the ugsplatform application file. Use the ugsplatform   for MAC or Linux.

This is a view of the UGS default windows

The windows can be dragged and dropped in different locations. We recommend organizing them as shown below.
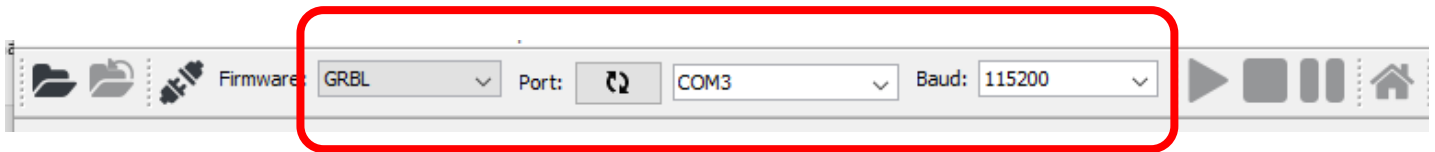

Don't worry, if you do not like the placement, you can always reset back to the Default Set Up by clicking the menu Windows, then Reset Windows.

Connecting UGS Platform to the CNC

Step 1

Check that the **Firmware** is set to **GRBL**, the **Port** is what you have when you looked at the Device manager, and the **Baud** rate is **115200** as shown.

Step 2

Click the connect button
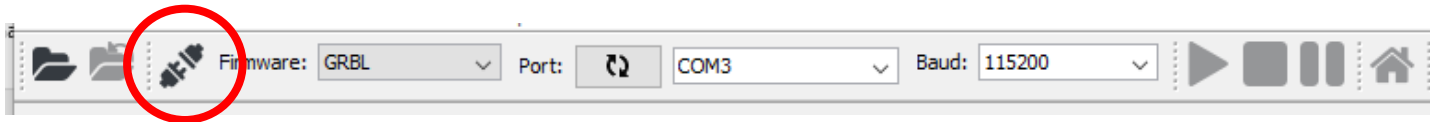
The UGS Screen should look as shown. Note, that the Controller state DRO (Digital Read Out) Window indicates ALARM Mode. This is normal and the mode will change once the CNC is successfully Homed. If your screen does not show the ALARM mode, then you need to troubleshoot and find the problem.

Step 3

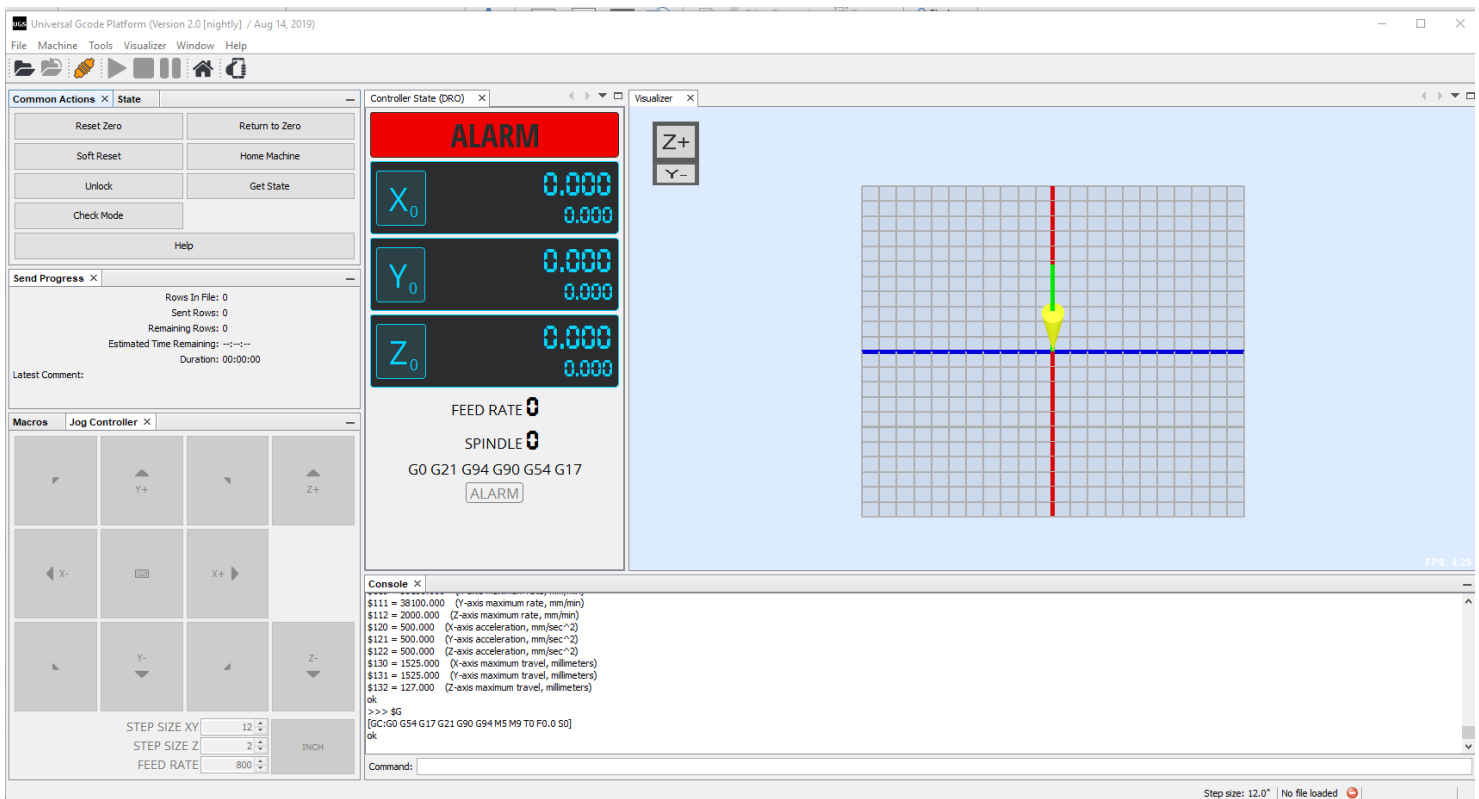Once you've connected you need to Home the CNC. This will drive all axes to the home switches and set soft limits in the firmware. Click the home button to Home the CNC.
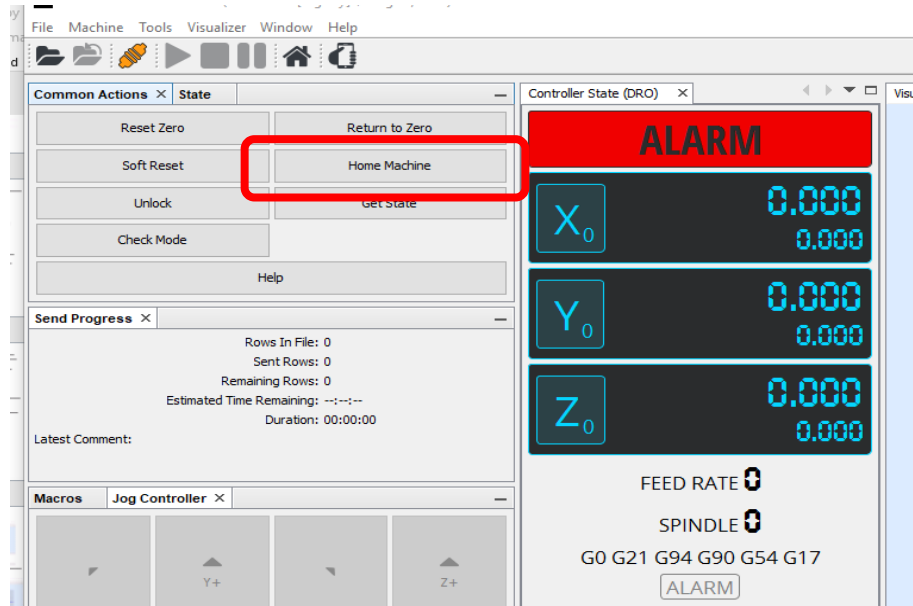
When the CNC moves to the home switches and the UGS Platform alarm mode is indicates IDLE, you have successfully homed your machine. If you are still in the ALARM mode then you will need to troubleshoot and find the problem.

Jogging the CNC in UGS Platform

Jogging means manually moving the Spidle along each axis to a desired position. You will need to understand and setup the Jog Controller window to successfully jog.

- The inch/mm button changes the jog units

- The XY Step Size controls how far the X or Y axis will travel each time an X or Y axis button is pressed

- The Z Step Size controls how far the Z axis will travel each time a Z axis button is pressed.

- The Feed rate is how Fast the axis will move the selected distance when a jog button is selected.

The jog function is useful for moving the Spindle in any or all of the axes to a desired position. This aids in setting up a project's starting position.

The values that are shown are good values to start with as you familiarize yourself with the processes.

NOTE: The CNC will not jog –X, -Y, or +Z from the home position. This is because the Z axis is up as far as it will travel and The X and Y are as far back as the can travel.

What you need to know when Creating a tool path (gcode) File

Once you have a CAD design you will need to create a tool path file. This file is typically referred to as a gcode file. This instruction is not to teach how to create a file. There is a learning curve when creating a tool path file. Here are the main things you will want to check before you save the gcode file.

- **What units are indicated?** The choice is inches or millimeters

- **What is the feed rate?** The feed rate set determines how fast the Spidle will travel in the X and Y axes. If the feed rate is too aggressive, it can break bits. If it is too slow it will burn the bits.

- **What plunge rate is set**? The plunge rate set determines how fast the Spindle travels in the Z axis. If this is too aggressive it can break bits. If it is too slow you will burn the bits.

- **What is the depth per pass**? The depth per pass can be set small to allow for faster feed rates. This typically will give the best results.

- **Where is the zero point set?** Usually, this is set to the lower left corner and at the top of the work piece surface. However, the zero point can also be set to the center of a workpiece or any other place desired. You must understand where the zero point is to successfully set up the project

- **Are the design dimensions within the travel limits of my CNC?** You will want to check to make sure the project will fit in the cutting area of the CNC.

- **What post processor is set?** BobsCNC uses *grbl1.1* firmware, your CAM software must be set up to use *grbl* specific gcodes. Look for a *grbl* setting in your CAM software. If you cannot find them use a "generic" setup.
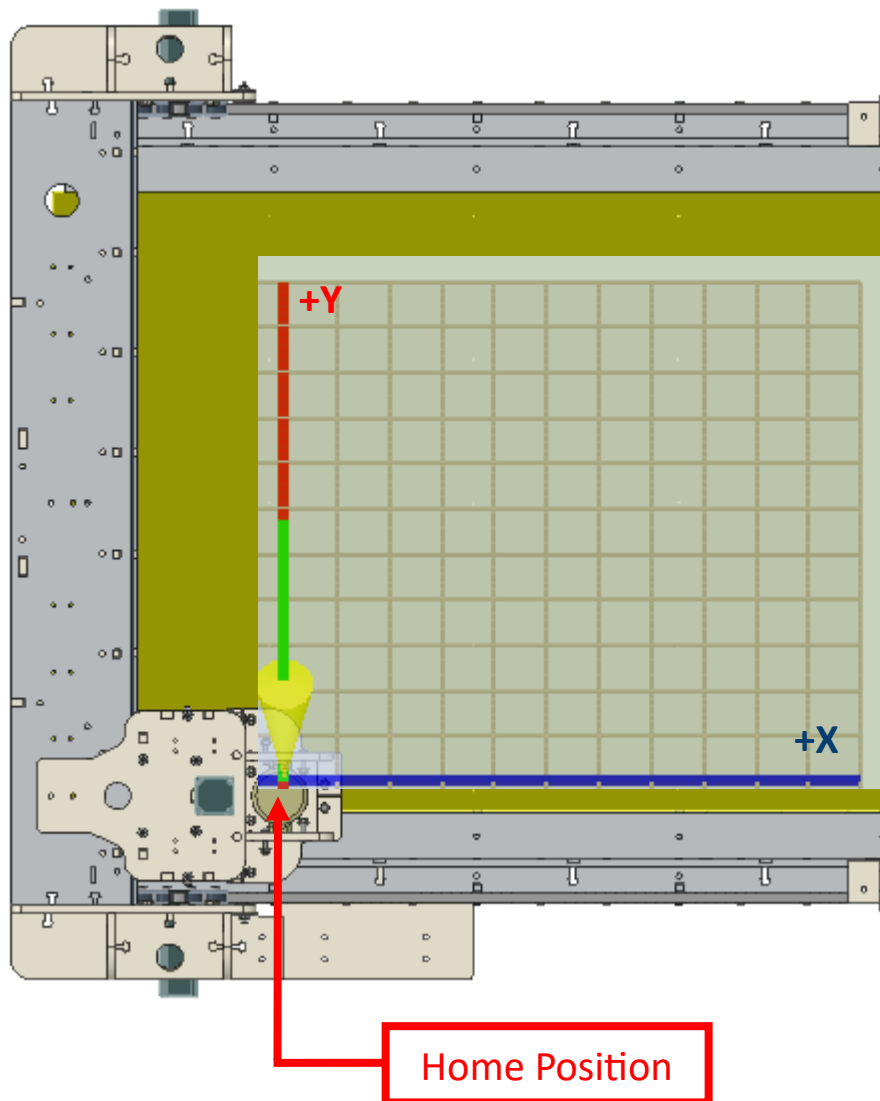
Understand the setup of your CNC

The BobsCNC Routers are setup with:

The X axis is the entire gantry moving and is horizontal on the computer screen.

The Y axis is the spindle moving across the gantry and is vertical on the computer screen.
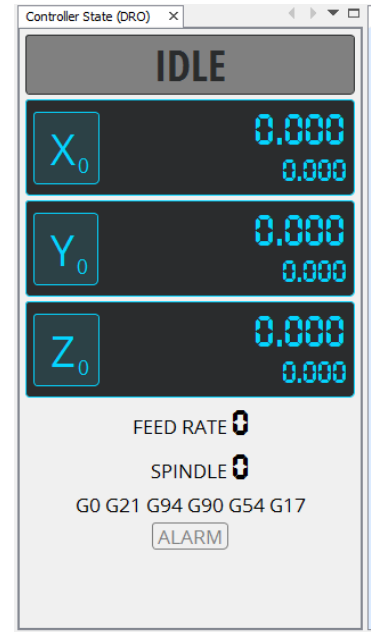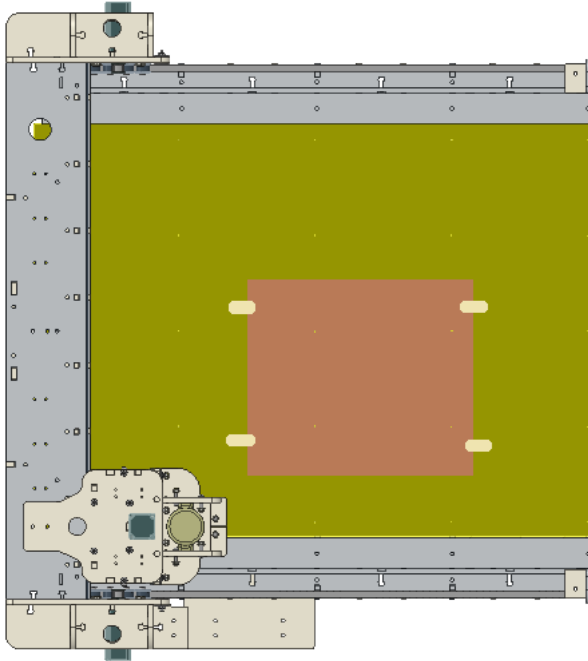
The Z axis moves the Spindle up and down and is in to or out of the computer screen.

The home position is the lower left corner of the CNC and will be the lower left corner of the computer screen.
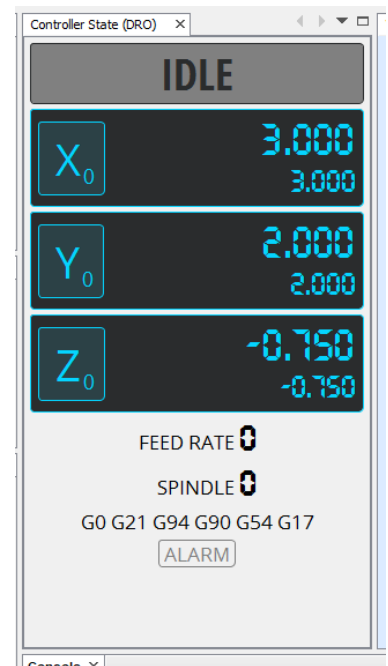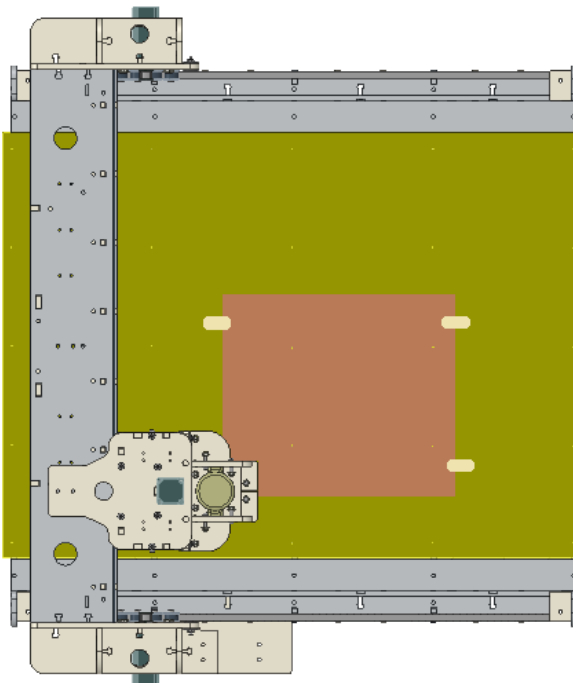

Home Position

Project Setup Process

Step 1 Clamp the workpiece within the cutting area of the CNC

Controller State (DRO) ×

IDLE

X_0 0.000 0.000

Y_0 0.000 0.000

Z_0 0.000 0.000

FEED RATE

SPINDLE

G0 G21 G94 G90 G54 G17

ALARM

Step 2 Jog to the start position on the workpiece which matches the start position of the tool path file.

Controller State (DRO) ×

IDLE

X_0 3.000 3.000

Y_0 2.000 2.000

Z_0 -0.750 -0.750

FEED RATE

SPINDLE

G0 G21 G94 G90 G54 G17

ALARM

Console ×

Step 3 Reset each axis to zero by clicking X0, Y0, and Z0.
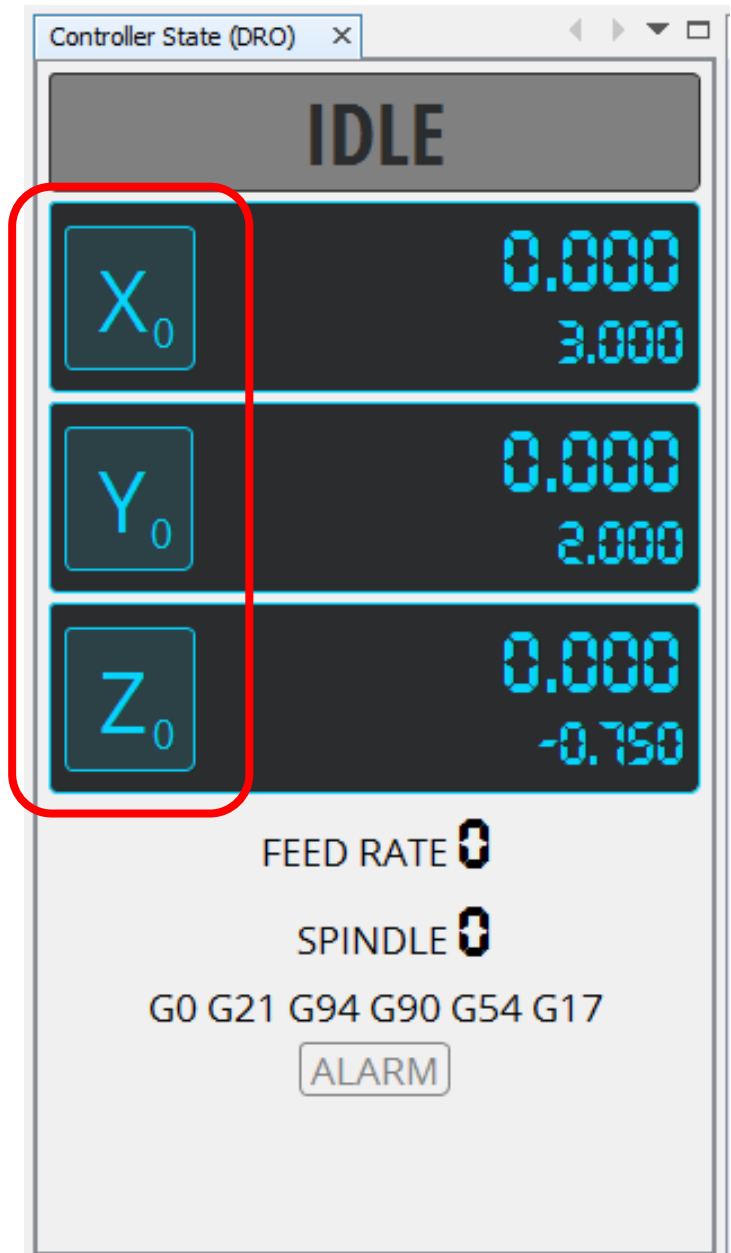
NOTE: The top larger number for each axis in the DRO (Digital Read Out) is matched to the zero point of the tool path file. The smaller numbers for each axis in the DRO is how far you have travelled from the home position.

Step 4 **Load the file, turn on the router, the dust collection system. Click play and the gcode file will start to run.**

Universal Gcode Platform (Version 2.0 [nightly] / Aug

File   Machine   Tools   Visualizer   Window   Help

Play, Stop, and Pause

Open Gcode File