Software Setup for BobsCNC

Contents

CNC Software Basics

Choosing the Right Software

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

1. CAD—Computer Aided Design software is the software you use to create designs. CAD can be as simple drawing program or a 3D model maker.

2. CAM—Computer Aided Machining 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.

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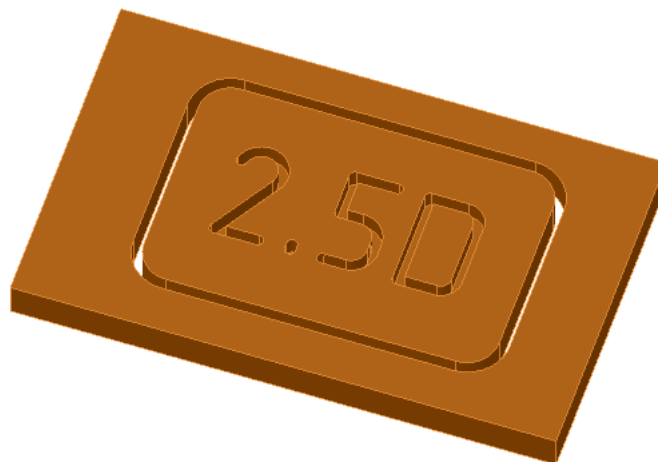
3D— projects include contour cuts where the CNC's Z axis is moving with the X and Y axes to create the 3D surfaces. Below is a simple 3D model.

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 Evolution Series CNC Router use an UNO microcontroller running *grbl1.1* to interpret the gcode files. When choosing a gcode sender you need one that is compatible with grbl1.1. We recommend the program, 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
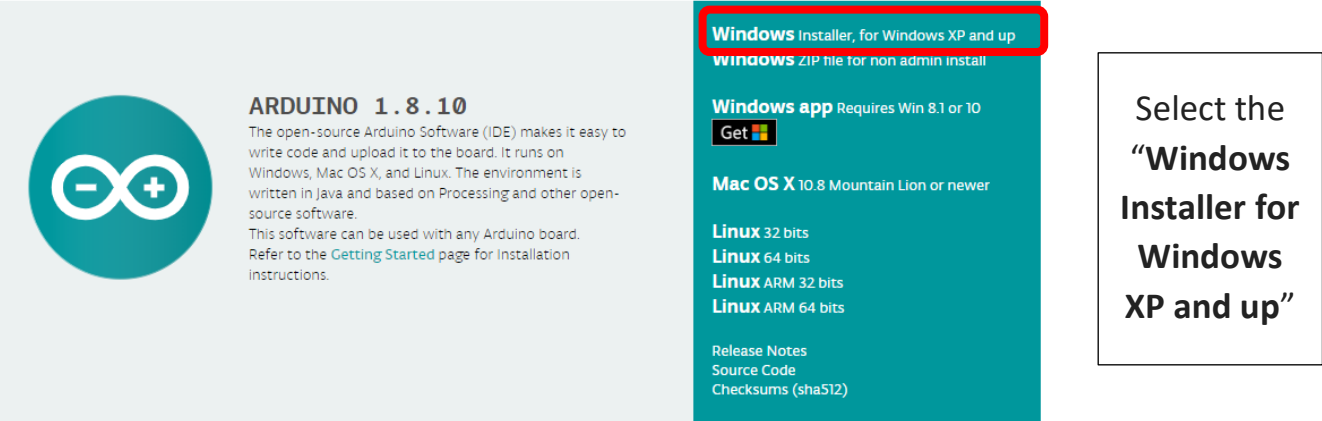- MeshCAMse
- Fusion 360
- dmap2gcode

Installing and Checking the Serial Driver and Installing UGS

What is a driver?

"A driver is a computer program that operates or controls a particular type of device that is attached to a computer. A driver provides a software interface to hardware devices, enabling operating systems and other computer programs to access hardware functions without needing to know precise details about the hardware being used" (https://en.wikipedia.org/wiki/Device_driver).

Step 1   Download and save the Arduino IDE software from:

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

Download the Arduino IDE

ARDUINO 1.8.10
The open-source Arduino Software (IDE) makes it easy to write code and upload it to the board. It runs on Windows, Mac OS X, and Linux. The environment is written in Java and based on Processing and other open-source software.
This software can be used with any Arduino board. Refer to the Getting Started page for Installation instructions.

Windows Installer, for Windows XP and up
Windows ZIP file for non admin install

Windows app Requires Win 8.1 or 10
Get

Mac OS X 10.8 Mountain Lion or newer

Linux 32 bits
Linux 64 bits
Linux ARM 32 bits
Linux ARM 64 bits

Release Notes
Source Code
Checksums (sha512)

Select the "**Windows Installer for Windows XP and up**"

You will be asked whether or not you want to contribute to help support the development of Arduino Software.

Contribute to the Arduino Software

Consider supporting the Arduino Software by contributing to its development. (US tax payers, please note this contribution is not tax deductible). Learn more on how your contribution will be used.

SINCE MARCH 2015, THE ARDUINO IDE HAS BEEN DOWNLOADED 38,606,221 TIMES. (IMPRESSIVE!) NO LONGER JUST FOR ARDUINO AND GENUINO BOARDS, HUNDREDS OF COMPANIES AROUND THE WORLD ARE USING THE IDE TO PROGRAM THEIR DEVICES, INCLUDING COMPATIBLES, CLONES, AND EVEN COUNTERFEITS. HELP ACCELERATE ITS DEVELOPMENT WITH A SMALL CONTRIBUTION! REMEMBER: OPEN SOURCE IS LOVE!

$3 $5 $10 $25 $50   OTHER

JUST DOWNLOAD   CONTRIBUTE & DOWNLOAD

Step 2   Use the USB to connect the controller to the computer and browse the Download folder and double clock the Arduino Software Installation file.

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 connected, open the Device Manager using the Search Textbox as shown.
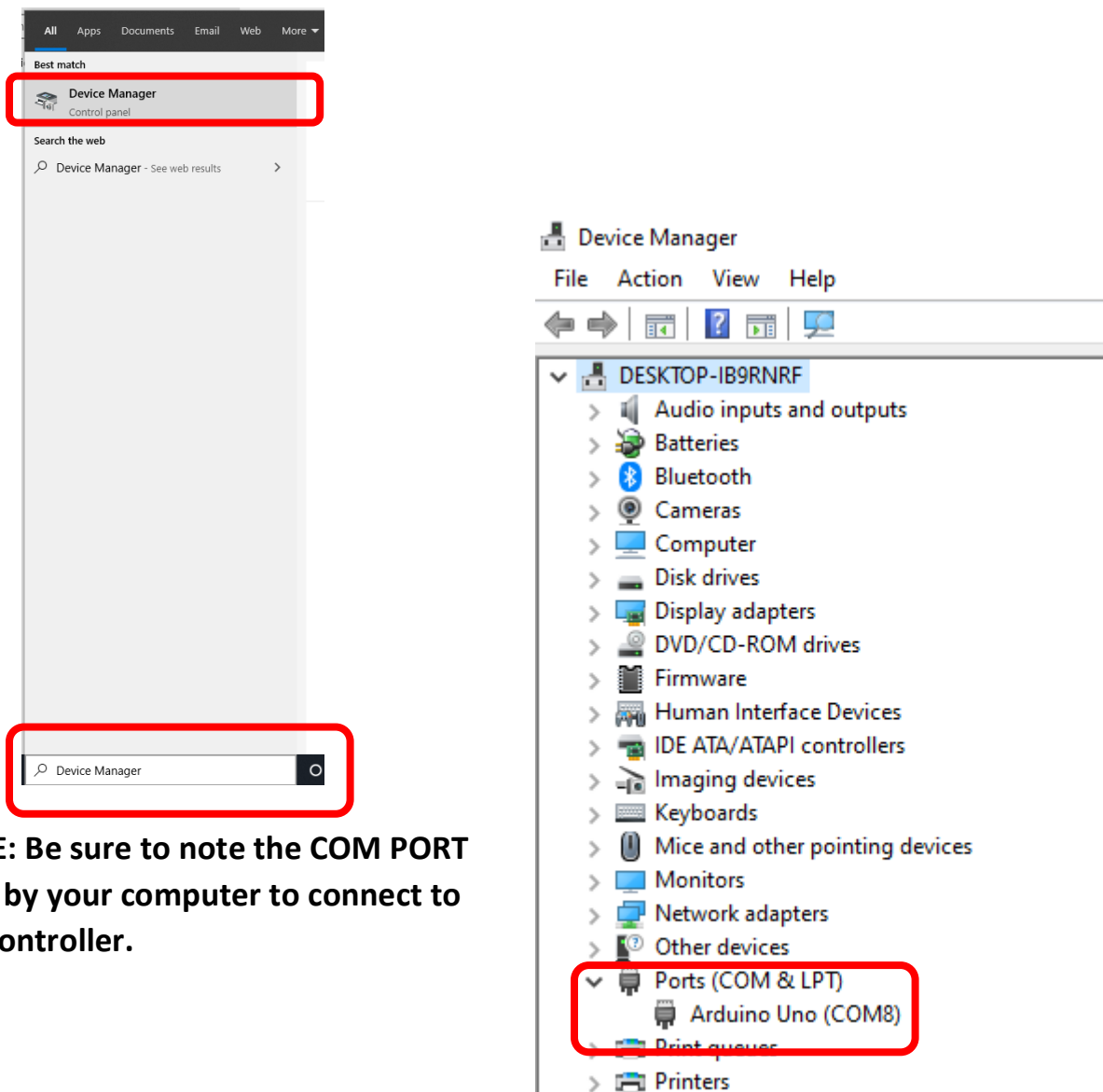
NOTE: Be sure to note the COM PORT used by your computer to connect to the controller.

Step 4 Browse to the UGS Download page and download the latest stable version of the **UGS Platform** for your operating system

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

Download

This is the latest release of UGS. For source code, nightly builds or older releases please visit github.

UGS Platform

The next generation platform-based interface.

Version	Operating system	Description
2.0.6	Windows	Windows version with bundled Java
2.0.6	MacOSX	MacOSX version with bundled Java
2.0.6	Linux	Linux version with bundled Java
2.0.6	Linux ARM	Linux ARM version with bundled Java. Can be used with RaspberryPi
2.0.6	All platforms	A generic package without Java which needs to be installed separately

UGS Classic

The classic UGS interface with slightly less features but with the same robust backend as the Platform edition.

Version	Operating system	Description
2.0.6	All platforms	A generic package without Java which needs to be installed separately

Step 5 Browse to the **DOWLOAD** folder and double-click the UGS

Platform Software zip file and unzip it to a location that you will remember.
NOTE: The version you unzip may be different than the one shown.

 ugsplatform-2.0.0-Aug.14.2019.zip

Step 6 Browse the **Bin** folder in the unzipped **UGS Platform** folder and

double clash the **UGS Platform** application file.

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

.lastModified
ugsplatform
ugsplatform.exe
ugsplatform64.exe

Getting to Know Universal Gcode Sender

What is Gcode Sender?

Universal Gcode Sender is the interface between your computer and your CNC controller. It sends gcode to the controller. The controller then makes calculations and sends instructions to the stepper motors or servo motors to move the spindle.

When you open UGS your screen will look like this. The screen consists of five windows
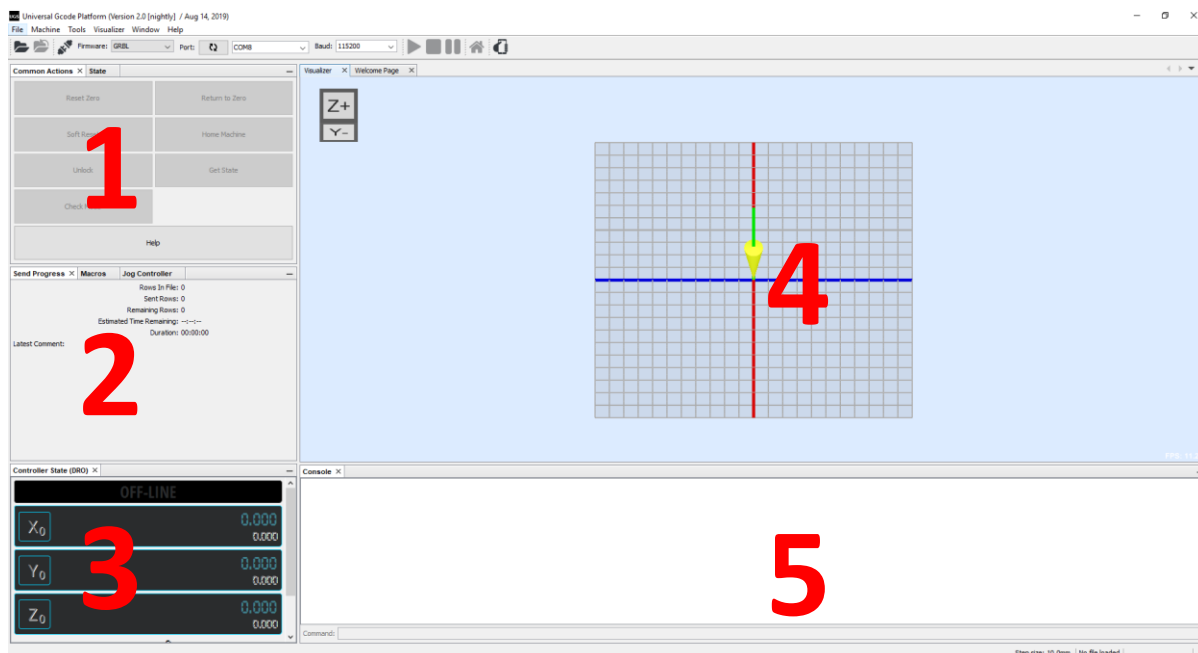
1. **Common Action/State Window**
2. **Jog Control Window**
3. **Digital Read Out Window (DRO)**

4. **Visualizer Window**
5. **Console Window w/ Command Box.**

For ease of use we recommend changing the layout of the UGS Screen from its default layout. The windows are easily moved by placing the cursor of your mouse on the title bar of the window you wish to move. Left click on the title bar and drag it to the preferred location.

As the window is moved you will notice that a highlighted box appears indicating a landing point has been found for the window. Continue moving the cursor and you will notice the size of the box will change. When the screen you selected is in the preferred location release the mouse. Notice the windows below have been moved in the location we recommend.
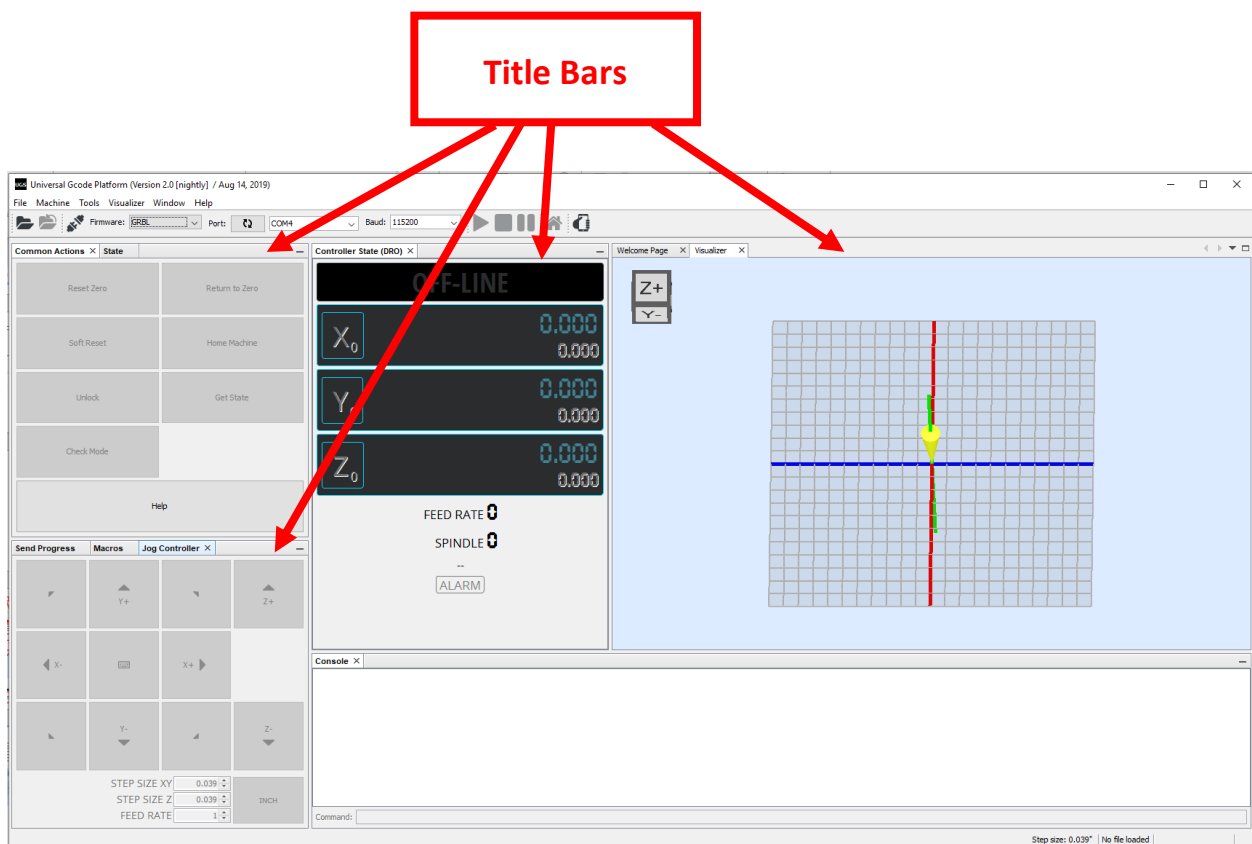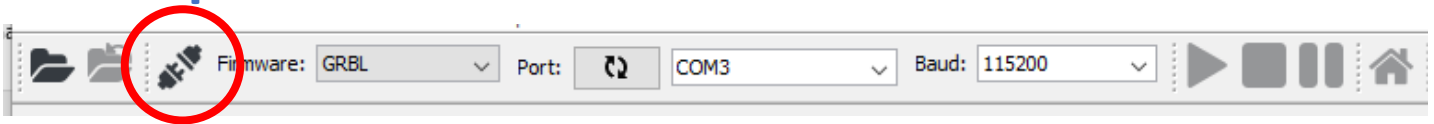
Connecting UGS Platform to the CNC

Step 1

Check that the **Firmware** is set to **GRBL**, the **Bin** folder in the unzipped **UGS Platform** folder and double click the **UGS Platform** application file.
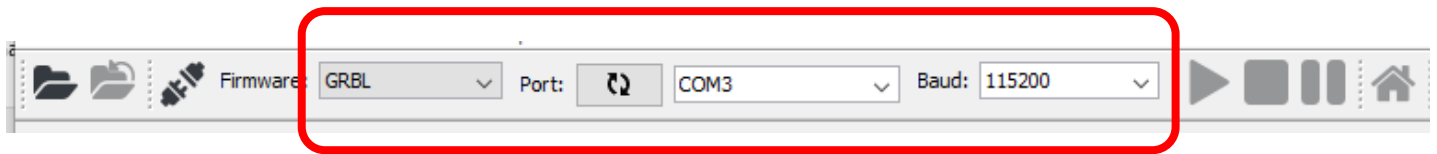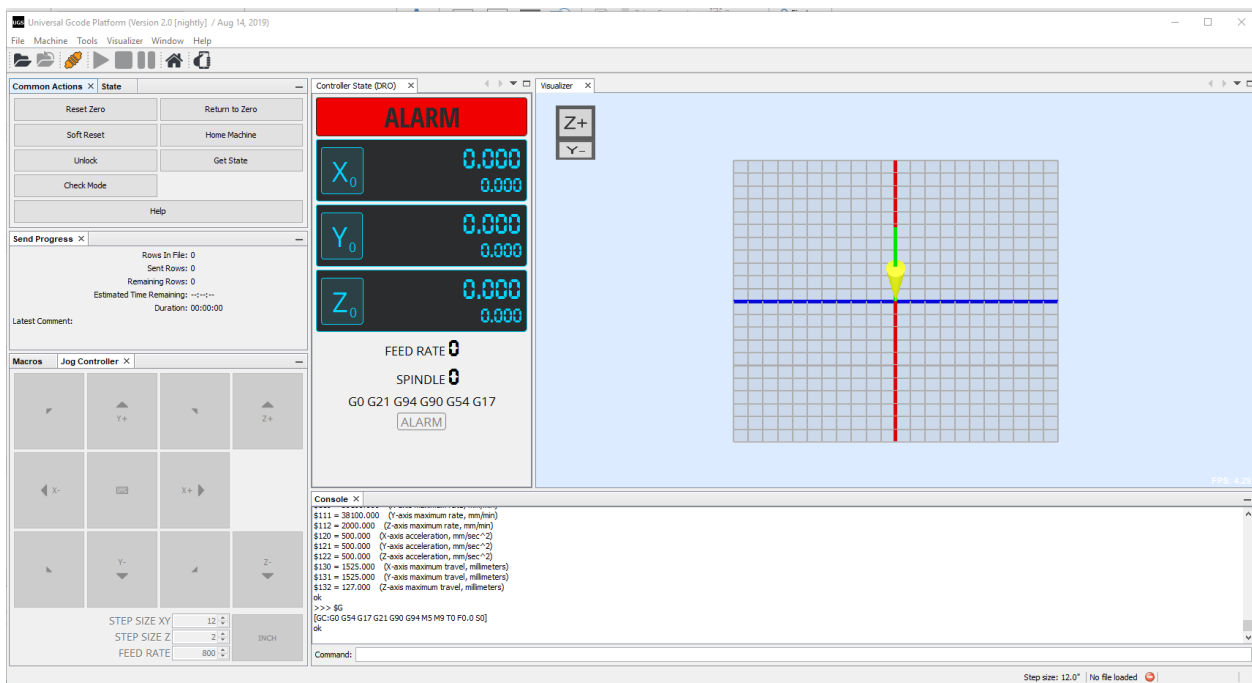

Step 2

Click the connect button.

The UGS Screen should look as shown. **Note**: the Controller State Digital Read Out Window (DRO) will indicate **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

After 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.
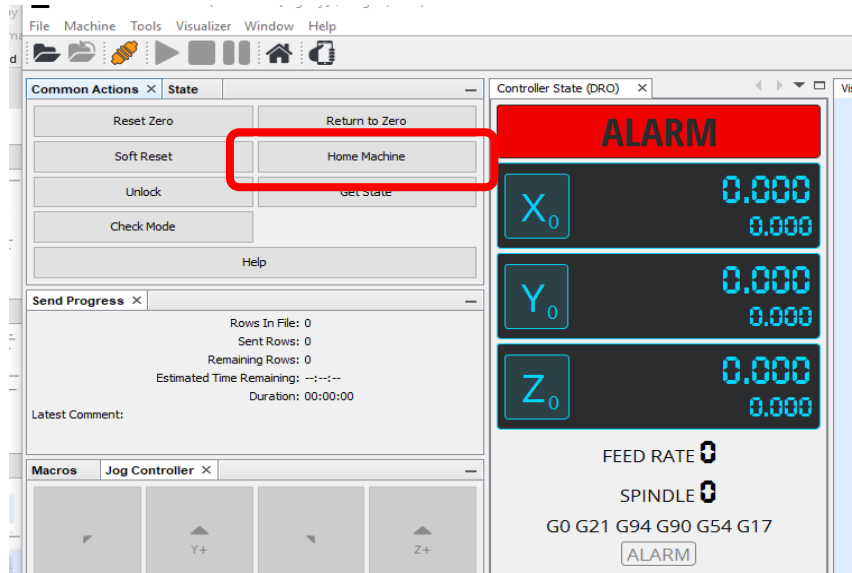
When the CNC moves to the home switches and the UGS Platform alarm mode is indicates **IDLE**, you have successfully homed your machine. If you are still in the ALARM mode then you will need to troubleshoot and find the problem.
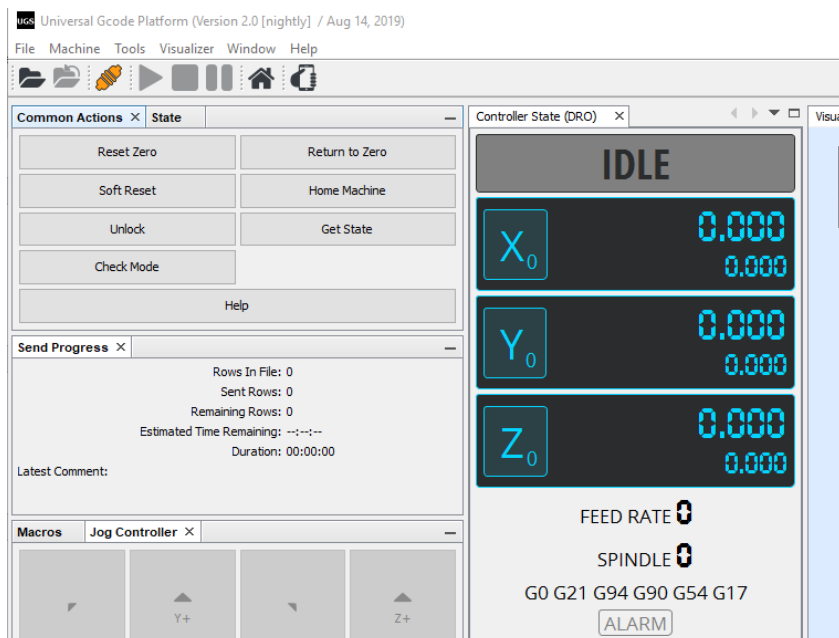
Jogging the CNC in UGS Platform

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.

What You Need to Know when Create a Tool Path and Save it as a 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?** 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.

Understanding the CNC Orientation

With a BobsCNC Router:

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.
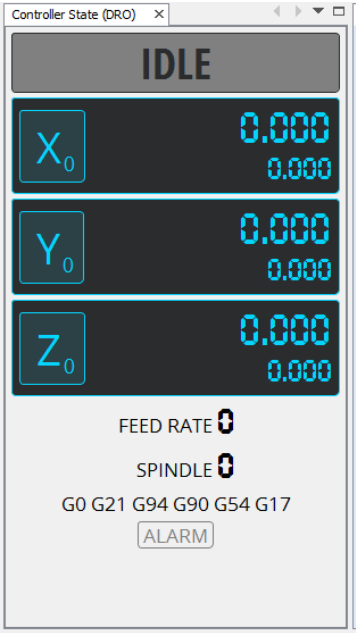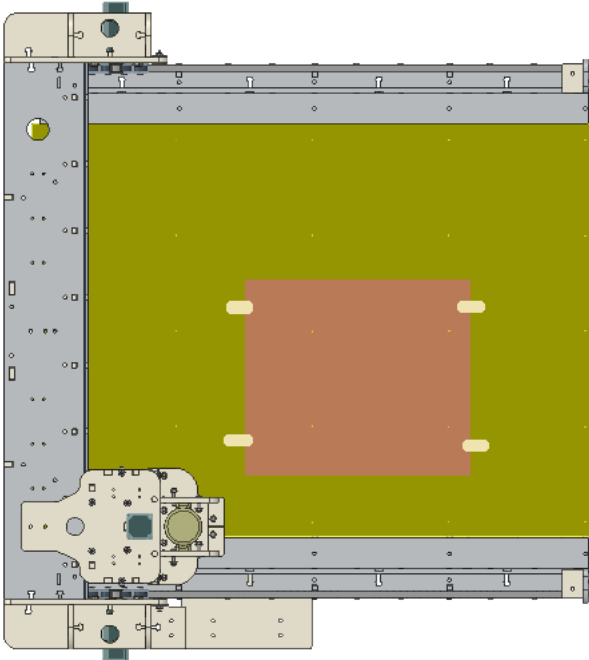
The Z axis moves the Spindle up and down and is in to or out of the computer screen.

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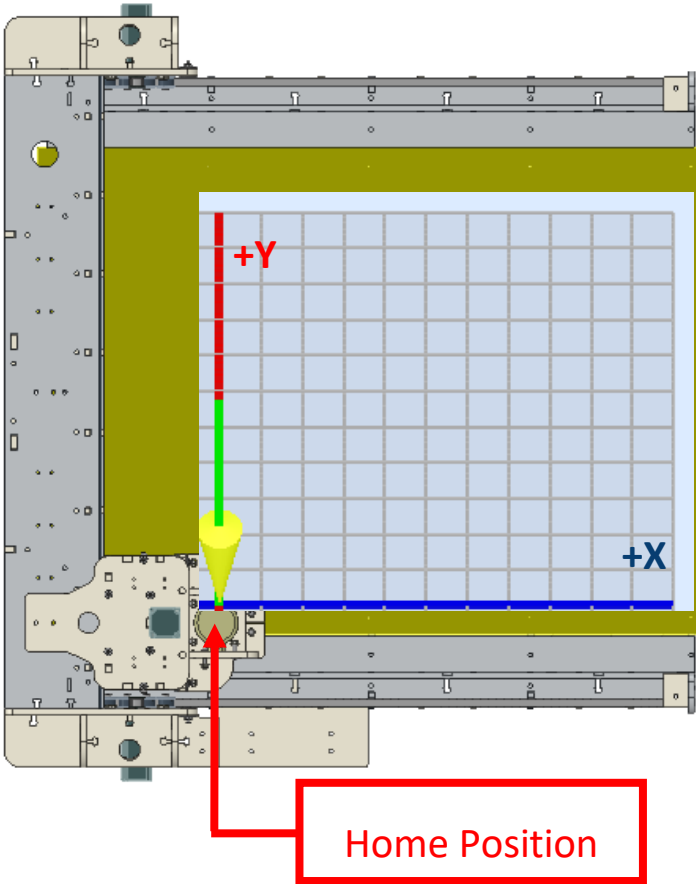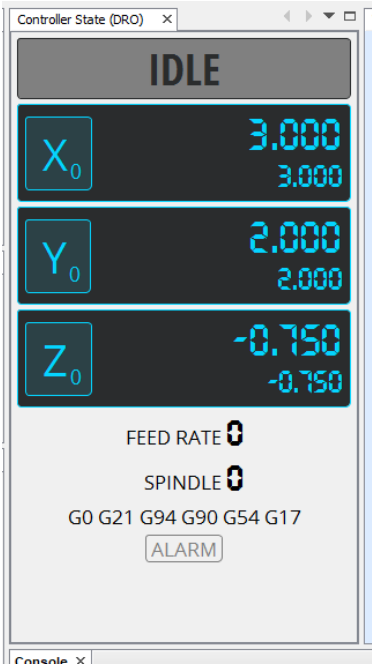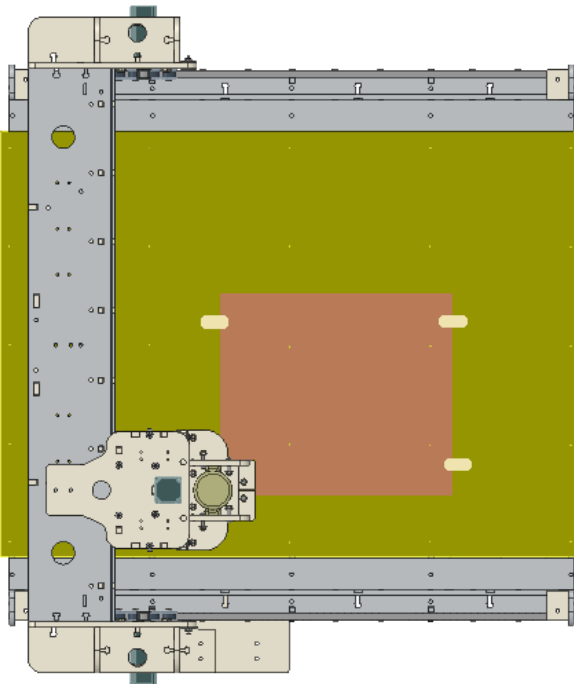
Project Setup

Step 1   Clamp the workpiece on the spoilboard.
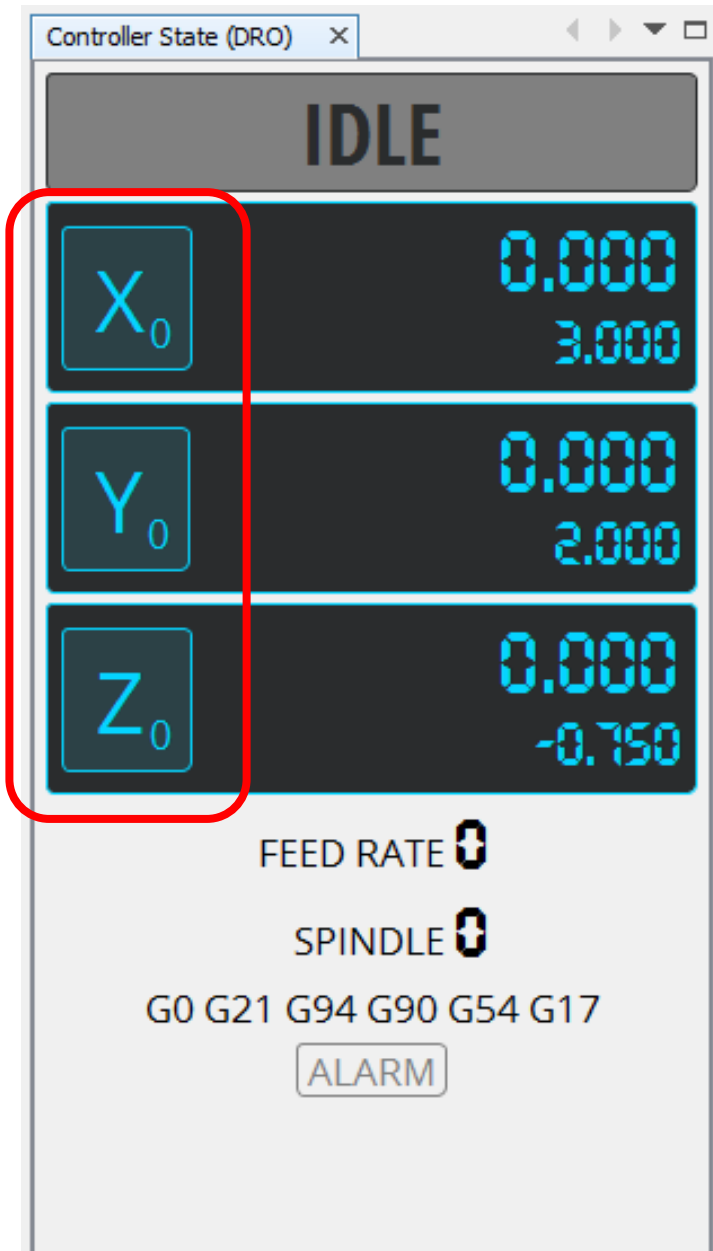

Step 2   Jog to the start position on the work piece which matches the start position of the tool path file.

Step 3   Reset each axis to zero by clicking the X0, Y0, and Z0 button.

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. Turn on the dust collection system. Click the Pay button and the gcode file will begin to run.

Play, Stop, and Pause

Open Gcode File