

# BASIC SENDER USERS MANUAL V0.0.36

## INTRODUCTION

Welcome to BobsCNC Basic SENDER. Basic SENDER is an easy to use and extremely stable gcode sender. It is built on the OpenBuilds® open source platform and was designed specifically for BobsCNC Router machines. Please forward questions, comments, and suggestions to [helpdesk@BobsCNC.com](mailto:helpdesk@BobsCNC.com).

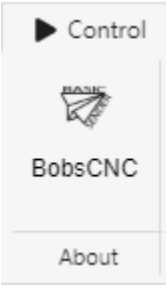
## DISCLAIMER

This hardware/software is provided "as is", and you use the hardware/software at your own risk. Under no circumstances shall any author be liable for direct, indirect, special, incidental, or consequential damages resulting from the use, misuse, or inability to use this hardware/software, even if the authors have been advised of the possibility of such damages.

## CONTROL

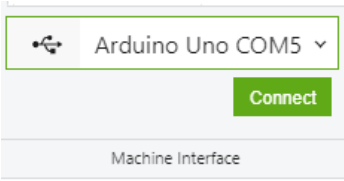
### MENU BAR

### ABOUT

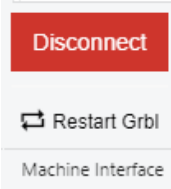


BobsCNC SENDER sends gcode to BobsCNC machines. It uploads BobsCNC firmware. It facilitates troubleshooting set up and runtime. It is based on the open source software created by OpenBuilds.com.

## MACHINE INTERFACE

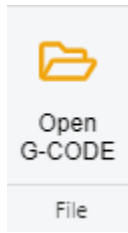


When you connect the controller on your BobsCNC router to your computer with a USB cord, the Status Bar will indicate the type of controller that is being used and the Com Port, which has been designated to link the computer and the controller.

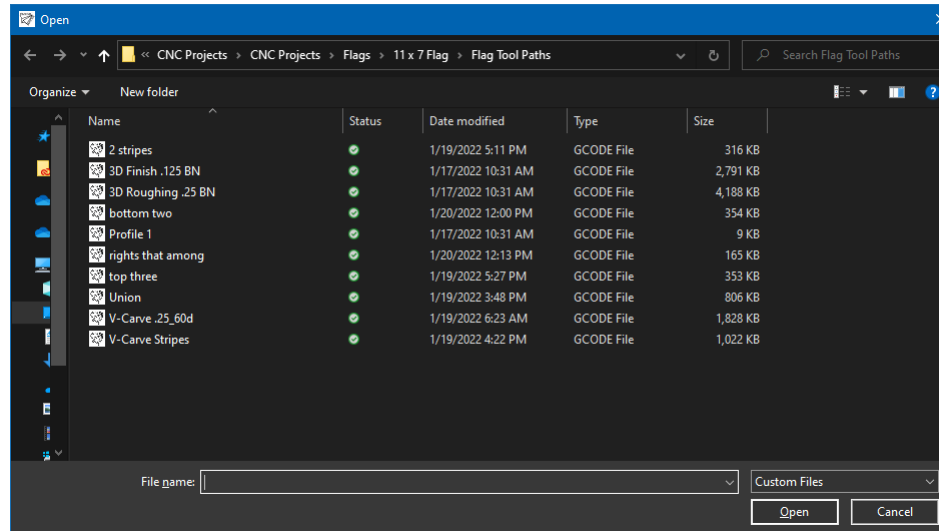


When you CLICK the Connect button, the display will change indicating your computer and controller are connected. I may be different than COM5.

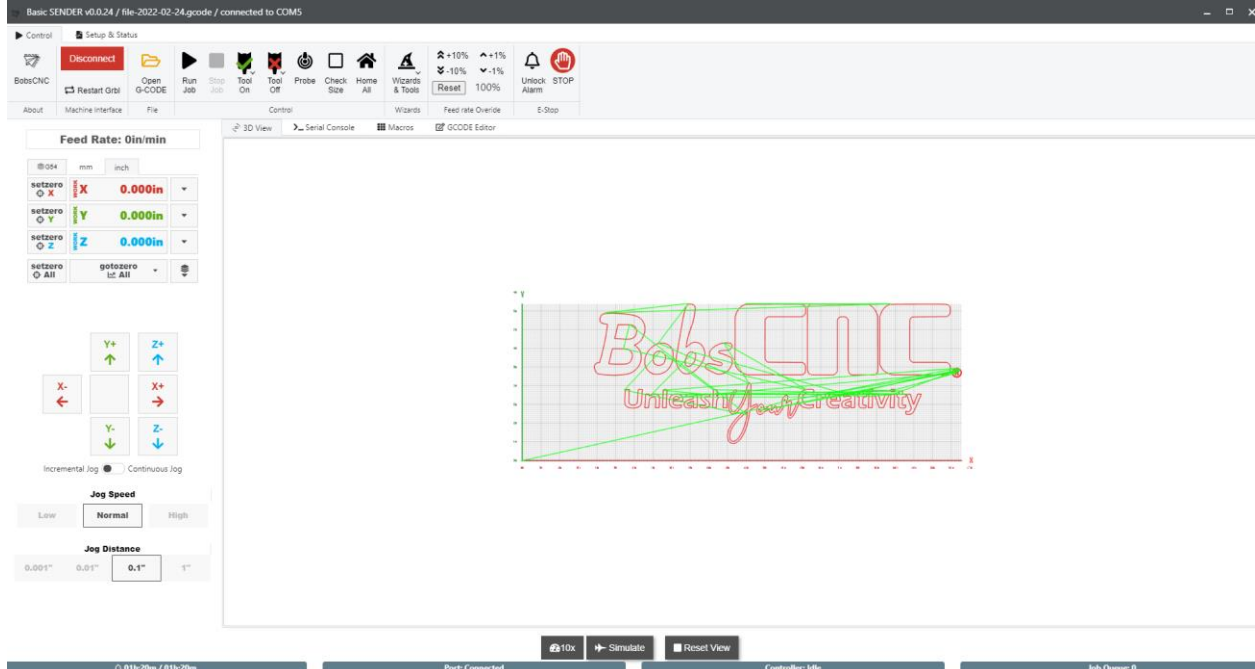
## FILE



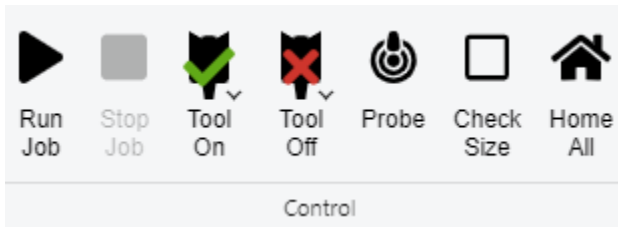
This section of the menu bar contains the Open G-Code icon. CLICK the icon, and you can search for the location where you have saved your gcode files as in the illustration below. Yours will vary.



After opening the file, it will be displayed in the 3D view as illustrated below.

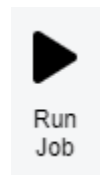


## CONTROL

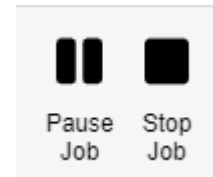


After the gcode is uploaded into Basic SENDER, the Control panel will look this.

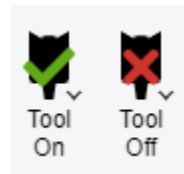
## RUN/PAUSE/STOP



When you CLICK the Run Job button, the gcode will be fed line by line to the controller. The icons will change giving you the option of either Pausing or Stopping your job.



## SPINDLE CONTROL

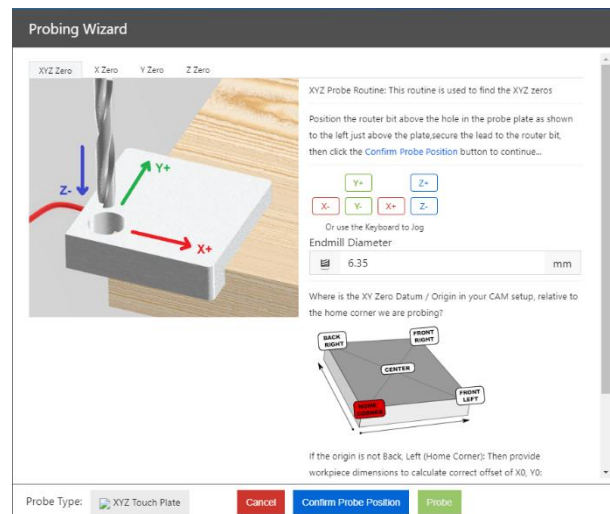
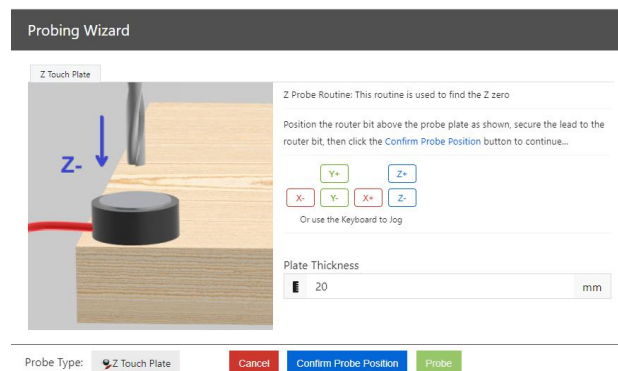


These icons are available for customers who have programmed an IOT switch to turn the Spindle on or off. Otherwise, the router can be manually turned on/off using the switch on the router.

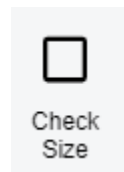
## PROBE



This icon opens the Probing Wizard which allows you to set up a Z Touch Plate and/or a XYZ Touch Place



## CHECK SIZE



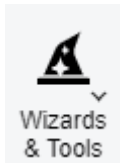
This icon initiates a process to validate that the gcode requirements of the uploaded file do not exceed the X, Y, and Z soft limits to keep from triggering an Alarm 2. It can also be used to diagnose an existing Alarm 2 issue.

## HOME MACHINE

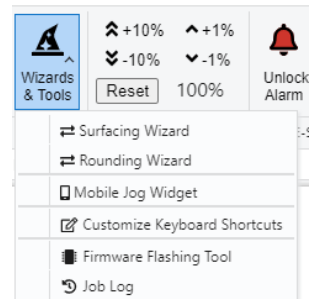


Homing your CNC is a process that must be performed every time the machine is turned on. Clicking the button will tell the controller to initiate the process of driving the spindle (router) to its home location. First the Z (router) carriage will rise to the Z home switch. After it triggers the Z home switch, it will descend 10 mm and slowly rise up and trigger the switch a second time. While this is happening, the Z carriage will travel across the Gantry to the Y home switch location. Finally, the Gantry will be driven to the X home switch locations.

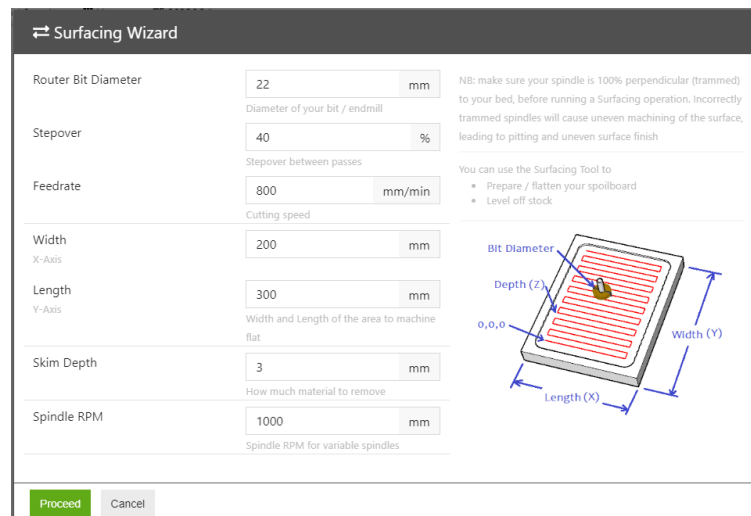
## WIZARDS & TOOLS



Clicking this icon will open up a number of helpful Wizard and Tool options.



## SURFACE WIZARD



The surfacing wizard helps you use your CNC Router like a planer to flatten the top surface of uneven stock or glued up panels.

You program in the diameter of the surfacing bit, the percentage of stepover, and the feed rate.

Next tell the wizard the width (X-axis) and length (Y-axis) of the material you are surfacing.

The Skim Depth indicates the total amount of material that will be removed in one pass.

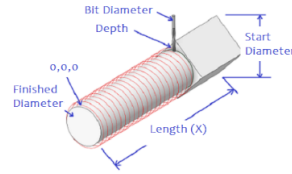
The Spindle RPM on hobby machines is normally controlled by using a manual setting on the router.

## ROUNDING WIZARD

⚙️
Rounding Wizard

Router Bit Diameter	<input type="text" value="6.35"/>	mm	NB: make sure your spindle is 100% perpendicular (trammed) to your centerline, before running a Rounding operation. Incorrectly trammed spindles will cause uneven machining of the surface, leading to pitting and uneven surface finish
	Diameter of your bit		
Stepover	<input type="text" value="40"/>	%	Stepover between passes
	Cutting speed		
Feedrate	<input type="text" value="1500"/>	mm/min	
Length <small>X-Axis</small>	<input type="text" value="200"/>	mm	
Start Diameter <small>A-Axis</small>	<input type="text" value="165"/>	mm	Length and Diameter of the area to machine round
	Length and Diameter of the area to machine round		
Finish Diameter <small>A-Axis</small>	<input type="text" value="150"/>	mm	Length and Diameter of the area to machine round
	Length and Diameter of the area to machine round		
Depth Per Pass	<input type="text" value="3"/>	mm	How much material to remove

Proceed
Cancel



The Rounding Wizard is used with the BobsCNC Revolution Rotary Axis machine. It enables you to transform square stock into round stock.

You program in the diameter of the router bit, the percentage of stepover, and the feed rate.

Next tell the wizard the length (X-axis) and start diameter (A-axis) and finish diameter of the material you are rounding.

The Depth Per Pass indicates the total amount of material that will be removed in one pass.

## MOBILE JOG WIDGET

📱
Jog Widget

You can use your mobile device as a handheld Jog interface. To get started, open a web-browser on your Mobile device and navigate to <http://192.168.254.72:3000/jog>

Close

The Mobile Jog Widget is a handy tool that allows you to use your mobile device as a pendant (a handheld Jog Interface).

## CUSTOM KEYBOARD SHORTCUTS

⌨️
Customize Keyboard Shortcuts

Click below to assign a new Keyboard Shortcut / combination to a function. Ctrl, Alt and Shift can be added to create combinations.

■ Stop / Abort	<input type="text" value="esc"/>
▶ Run / ⏸ Pause	<input type="text" value="space"/>
↻ Setzero XYZ	<input type="text" value="insert"/>
🏠 Goto XYZ Zero	<input type="text" value="del"/>
🔔 Unlock Alarm	<input type="text" value="end"/>
🏠 Home	<input type="text" value="home"/>
← Jog X-	<input type="text" value="left"/>
→ Jog X+	<input type="text" value="right"/>
↓ Jog Y-	<input type="text" value="down"/>
↑ Jog Y+	<input type="text" value="up"/>
⏴ Jog Z-	<input type="text" value="pagedown"/>
⏵ Jog Z+	<input type="text" value="pageup"/>
↶ Jog A-	<input type="text" value="undefined"/>
↷ Jog A+	<input type="text" value="undefined"/>

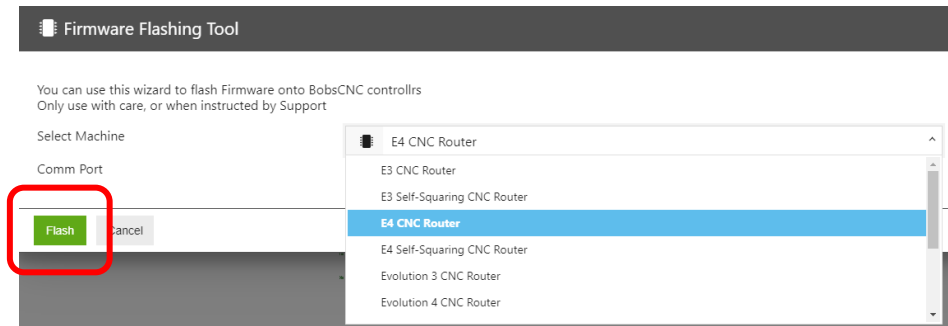
The Custom Keyboard Tool lets you create keyboard shortcuts to make using the Gcode Sender even more efficient.

<span>⏴ Decrease Step Size</span> <small>For Incremental Jogging</small>	<input type="text" value="-"/>
<span>⏵ Increase Step Size</span> <small>For Incremental Jogging</small>	<input type="text" value="+"/>
<span>⏴ Decrease Jog Speed</span>	<input type="text" value="0"/>
<span>⏵ Increase Jog Speed</span>	<input type="text" value="."/>
<span>⏴ Incremental Jog Mode</span>	<input type="text" value="/"/>
<span>⏵ Continuous Jog Mode</span>	<input type="text" value="*"/>
<span>⏴ Increase Feed Override</span>	<input type="text" value="q"/>
<span>⏵ Decrease Feed Override</span>	<input type="text" value="a"/>

Save and apply
Cancel

---

## FIRMWARE FLASHING TOOL



In the event that your firmware values are changed or corrupted, this wizard will automatically reflash the firmware settings on the controller. You simply have to click the drop down menu and highlight the name of BobsCNC machine you are using and CLICK the Flash button (outlined in red above).

---

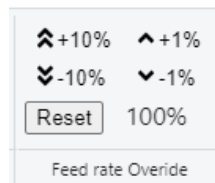
## JOB LOG

Status	Date	Name	Time
✖	Thu Feb 24 2022 09:35 AM	File2022-02-24.gcode	01:10:14 (Estimated) 01:10:14 (Estimated)
✖	Thu Feb 24 2022 09:35 AM	File2022-02-24.gcode	01:10:14 (Estimated) 01:10:14 (Estimated)
✖	Thu Feb 24 2022 09:35 AM	File2022-02-24.gcode	01:10:14 (Estimated) 01:10:14 (Estimated)
✖	Thu Feb 24 2022 09:35 AM	2-0101.gcode	01:10:14 (Estimated) 01:10:14 (Estimated)

The job log lets you track the amount of time it takes to execute each gcode file comparing it with estimated run times. This information can help you determine which tool paths require the most machine time revealing opportunities for optimization of CAM tool settings (e.g., increase of bit diameter, feed rate, or rapids setting).

---

## FEED RATE OVERRIDE



This feature allows you to increase or decrease the feed rate while a gcode file is running.

---

## E-STOP



CLICK this icon to stop Gcode Sender and abort the machining process

---

## JOG PANEL

---

## FEED RATE DISPLAY

**Feed Rate: 0in/min**

THIS DISPLAYS THE SPEED THE SPINDLE IS MOVING IN ANY PARTICULAR AXIS.

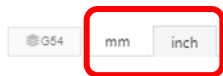
---

## DIGITAL READOUT (DRO)



---

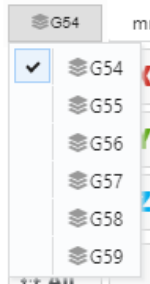
## SET UNIT DISPLAY



This allows you to tell Gcode Sender if your gcode is in mm or inches

---

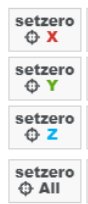
## SET WORK OFFSETS



This feature allows you to zero multiple projects on your machine. Most hobby cnc routers only use the G54 offset because they are only machining one project at a time.

---

## SET ZERO



This feature is used when zeroing your work piece. You can either set the zero of each axis individually or set them all at once ("setzero All").

---

## GO TO ZERO



This button will tell the machine to return the spindle to the work zero location

---

## SET PREDEFINED POSITIONS

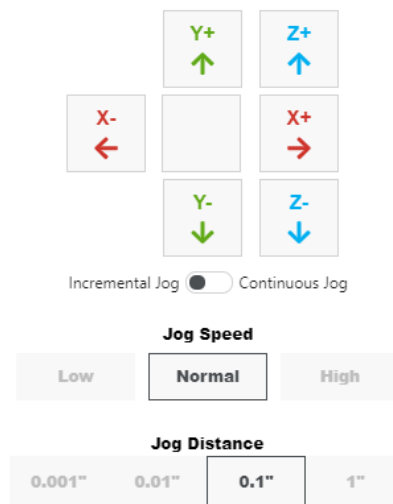


This is a feature that should only be used by advanced users. For more information see the link below:

[https://linuxcnc.org/docs/2.6/html/gcode/gcode.html#sec:G30-G30\\_1](https://linuxcnc.org/docs/2.6/html/gcode/gcode.html#sec:G30-G30_1).

---

JOG INPUT IS A CONTROL PANEL OF DIRECTIONAL KEYS AND BUTTONS TO SET THE DIRECTION, SPEED, AND DIRECTION OF THE JOG FUNCTION.



---

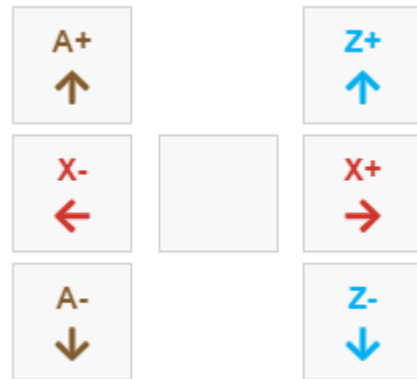
## JOG BUTTONS



### Standard Configuration (X, Y, Z)



### Rotary Configuration (X, A, Z)



BobsCNC Routers operate in the Right Hand Coordinate System therefore in reference to the Home Position (X0, Y0, Z0) the X and Y arrow buttons move the spindle away from their home positions in a positive direction, while the Z button moves the spindle away from its home position in a negative direction.

The Y Axis (Revolution Rotary Axis machine) Looking from the chuck toward the tail piece when the machine rotates counterclockwise, it is moving in a positive direction and clockwise in the negative direction.

---

### JOG FEED RATE

**Jog Speed**

Low   **Normal**   High

Low = 12 inches per minute.  
Normal = 197 inches per minute.  
High = 394 inches per minute

---

### JOG DISTANCE

There are two settings Incremental Jog or Continuous Jog

Incremental Jog  Continuous Jog

**Jog Speed**

Low   **Normal**   High

**Jog Distance**

0.001"   0.01"   **0.1"**   1"

The Incremental Jog setting allow you to move the spindle in predetermined distances from .001" to 1" at the speed chosen. This setting should be used for the final movement of the spindle when zeroing the bit.

Incremental Jog  Continuous Jog

### Jog Speed

Low

Normal

High

Continuous Jog moves the spindle in a continuous direction at the chosen speed as long as the button is activated.

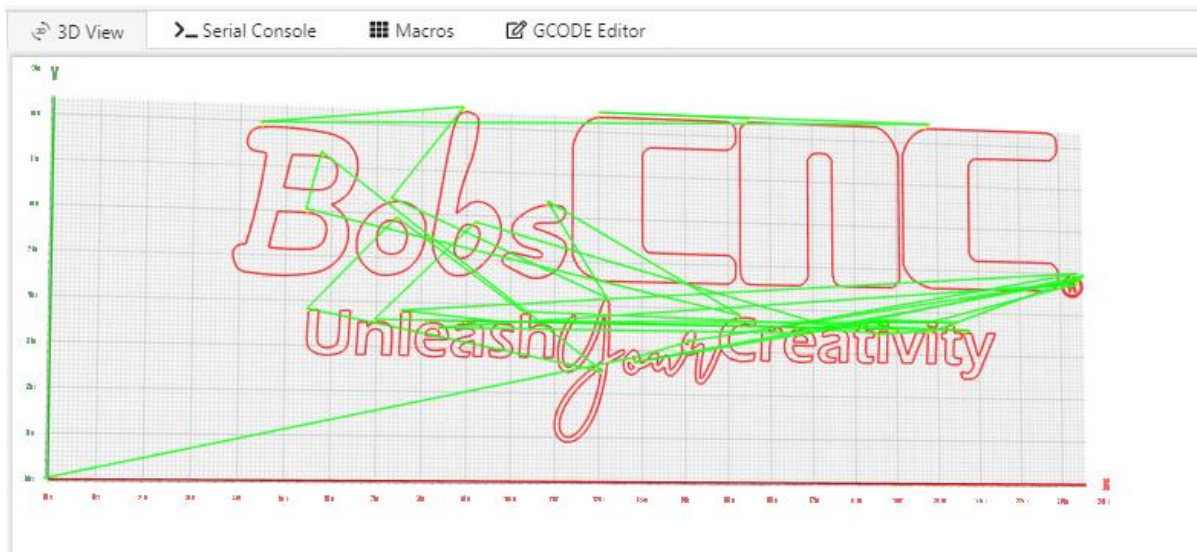
## DISPLAY PANEL

The Display Panel contains the 3D View, Serial Console, Macros, and GCODE Editor

### 3D VIEW

Allows you to zoom in, pan, and rotate the virtual representation of the tool path. The left mouse button selects the image. Spinning the scroll wheel lets you zoom in and out. The right mouse button allows you to pan image across the screen. Depressing the scroll wheel lets you rotate the image in 3D.

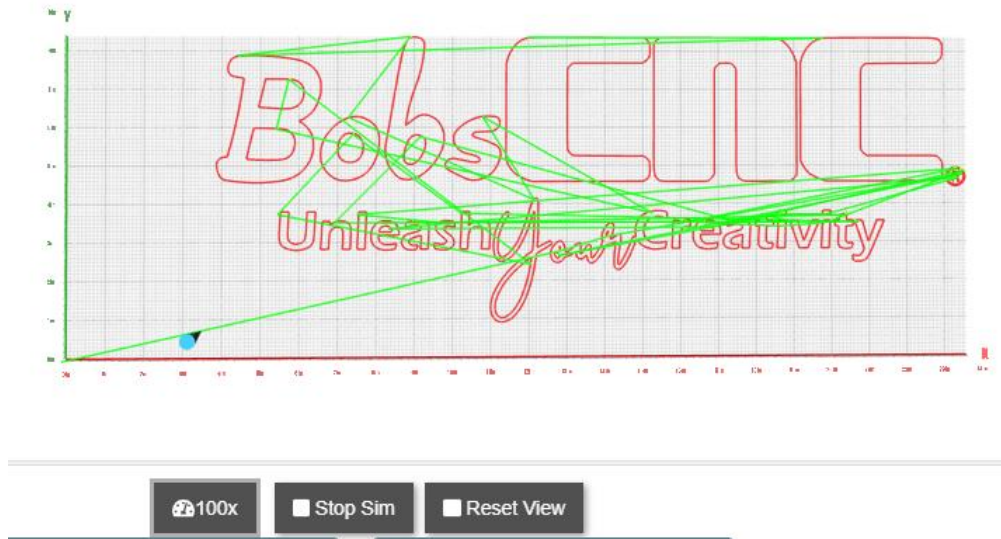
### DISPLAY WINDOW



---

## SIMULATION PANEL

The simulation panel allows you to run your gcode file in virtual reality.



The simulation can be modified to play at .1x to 1000x's the actual feed rate. The simulation can be stopped at any time during its playback and the screen can be reset.

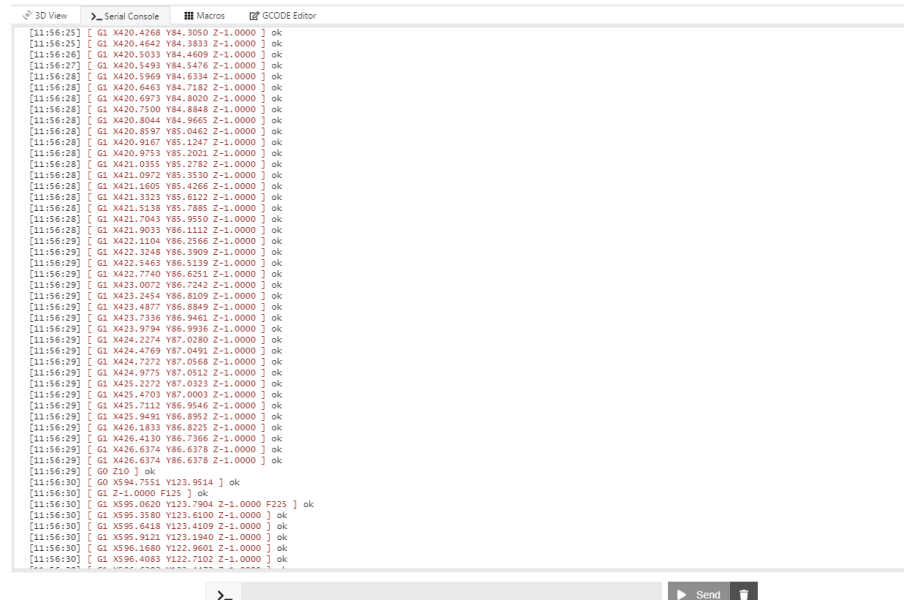
---

## SERIAL CONSOLE

The serial console let you to see the information that is being transmitted via the USB connection

---

## DISPLAY WINDOW

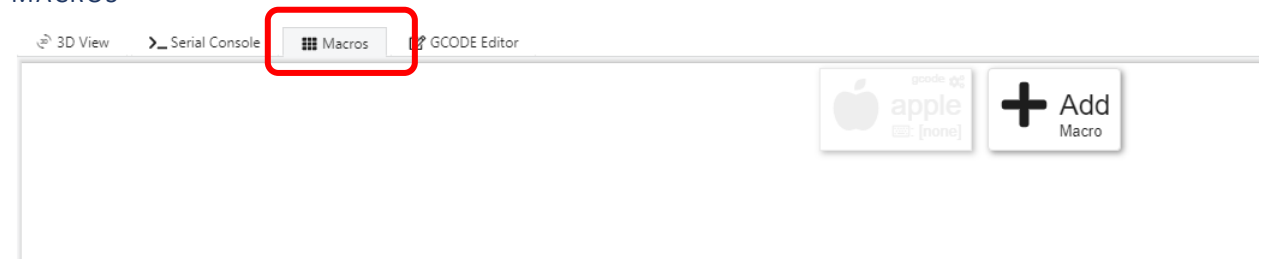


## COMMAND INPUT

This where you can insert gcode commands.

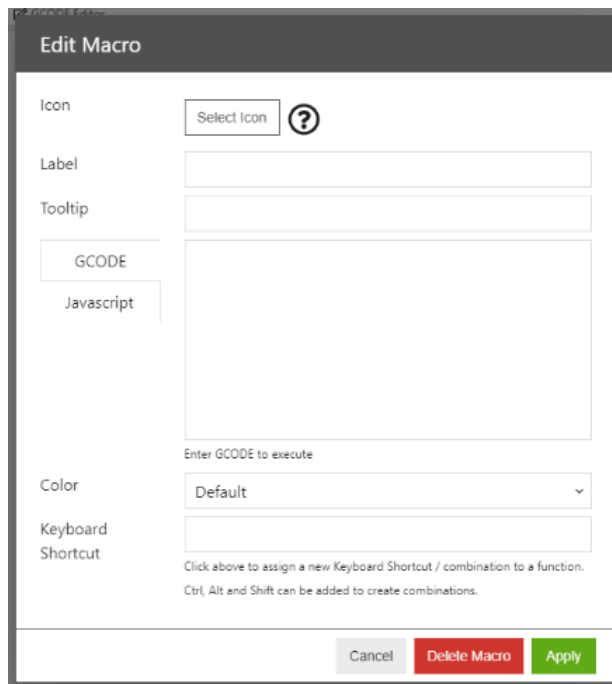


## MACROS



## DISPLAY WINDOW

This window displays all the macros that you have created.



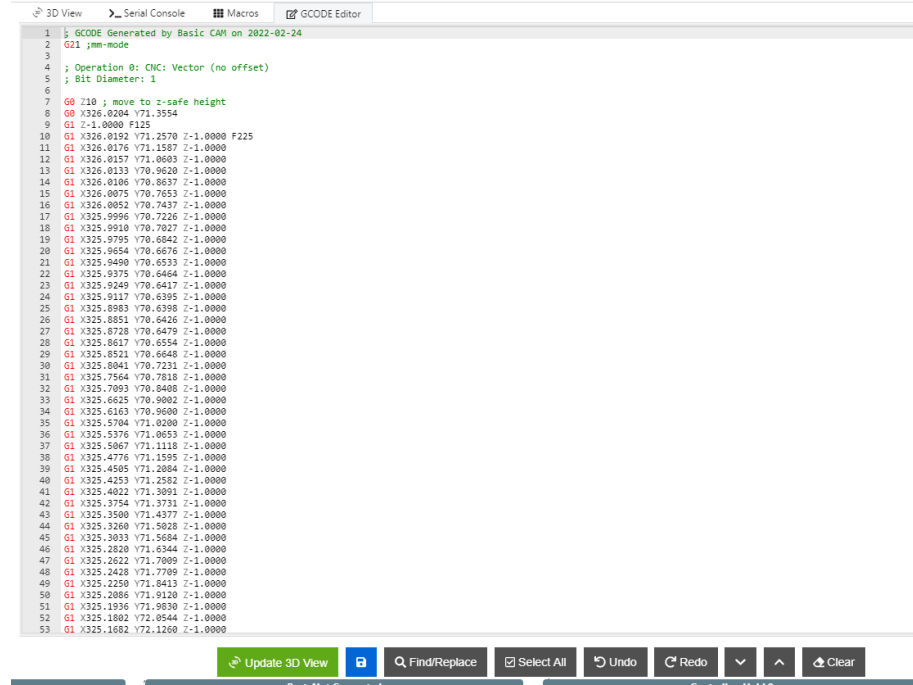
This is the panel that allows to:

- choose and icon to represent your macro
- Create a label for the macro
- Show the GCODE command to execute
- Show the Keyboard Shortcut used to access the macro
- Delete unwanted macros.

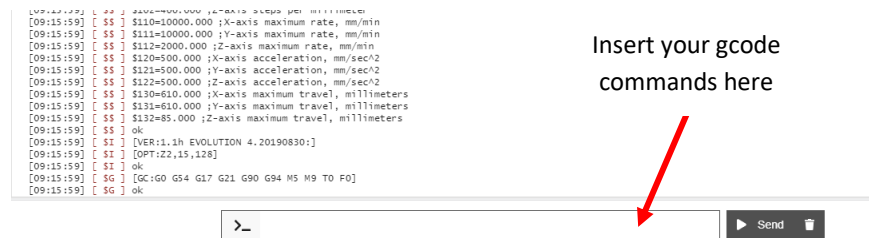
## GCODE EDITOR

This feature allows to so edit, add, remove, copy and paste lines of gcode.

## DISPLAY WINDOW

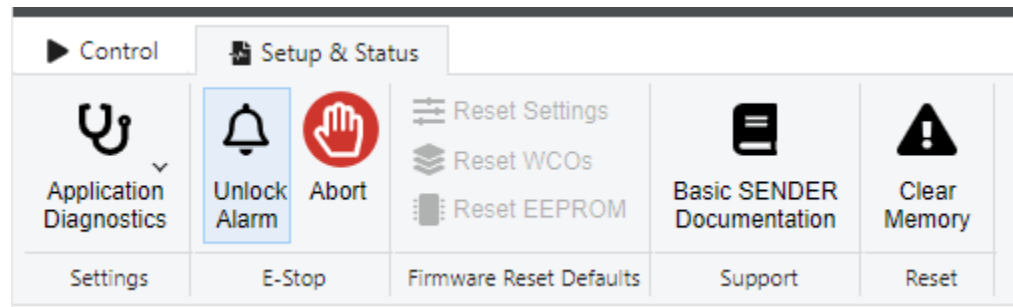


## EDIT GCODE PANEL



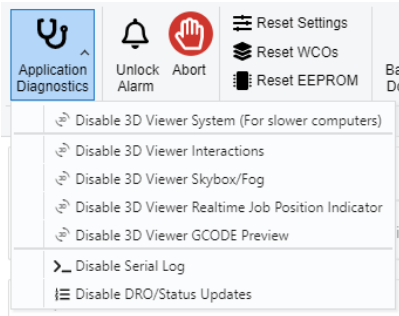
## SETUP

## MENU BAR



---

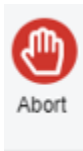
## APPLICATION DIAGNOSTICS BUTTON



Select or deselect these functions to tailor your interface experience.

---

## E-STOP

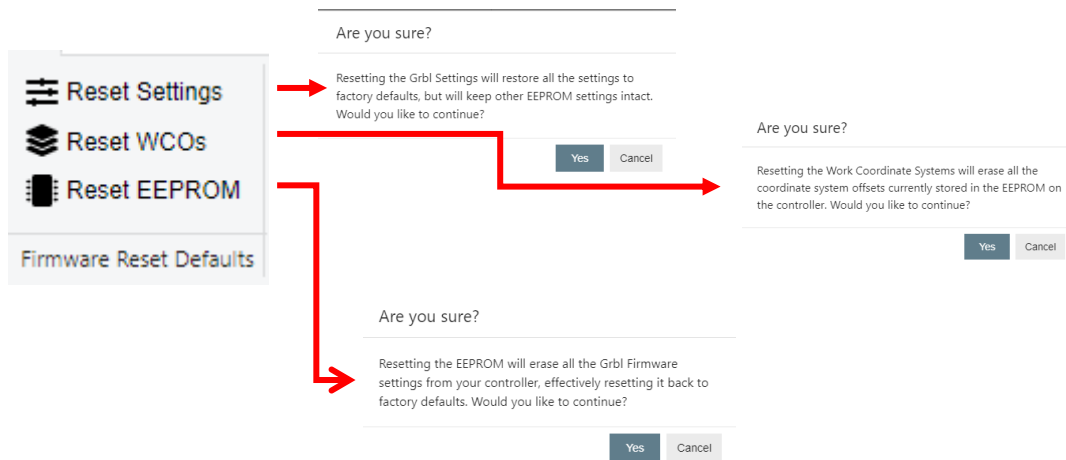


CLICK the E-Stop to stop transmitting the gcode file.

---

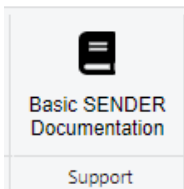
## FIRMWARE RESET DEFAULTS

In the event the firmware setting on the controller have been altered or corrupted they can be easily restored with this feature.



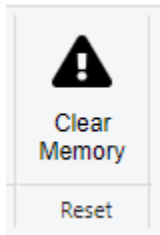
---

## BASIC-SENDER DOCUMENTATION BUTTON




This opens the BobsCNC Documentation page.

## CLEAR MEMORY BUTTON




This will set all values back to the defaults.

## RECOVERY

 Recovery: Last line ran found from earlier job run: NA

When you run a gcode file this will keep track of the last line ran if the run is stopped or you loose power.


## AXIS DISPLAY

 DRO Axis Display

Show X Axis  Show Y Axis  Show Z Axis  Show A Axis

This feature allows you to choose which AXES are displayed. X, Y, Z for the Evolution and KL Series CNC Routers or X, A, Z for the Revolution Rotary Axis CNC Router.

## INPUTS & HOME SWITCHES

 Inputs / Home Switches

PIN	Status
X-Limit	ON
Y-Limit	ON
Z-Limit	ON
Probe	OFF
Door Sensor	OFF
Buttons	START-OFF HOLD/DOOR-OFF RST-OFF

This gives you a quick way to verify the Home switches and other input signals

## COMMUNICATIONS

Communications	
Value	Status
Installed Version	v0.0.24
Backend Queue Blocked	Ready
Connection Status	Connected
Connected To	COM5
Serial Queue	0
Websocket Status	Connected

This gives you a quick way to verify potential communications issues with your controller.

## JOG SPEED SETUP

Jog Speed Setup		
Description	Value	
Slow Jog Speed (%)	1	%
Medium Jog Speed (%)	20	%
Fast Jog Speed (%)	60	%
Rapid Jog Speed (%)	100	%

This gives you a quick way to edit your jog feed rates based on the 100% rapid rate set by your controller

## JOG DISTANCE SETUP

Jog Distance Setup		
Description	Value	
Tiny (mm)	0.03	mm
Small (mm)	0.1	mm
Medium (mm)	3	mm
Large (mm)	25	mm
Tiny (inch)	0.001	inch
Small (inch)	0.01	inch
Medium (inch)	0.1	inch
Large (inch)	1	inch



This gives you a quick way to edit your jog distances both for inches and mm.

## CHANGE LOG

### Changelog

- v.0.0.35: Update jog pendant to hide axis config and set jog rates
- v.0.0.34: Update autoupdate
- v.0.0.33: Update code for startup window.
- v.0.0.32: Update the code to look for updates
- v.0.0.31: Add inputs to customize the feed rate and jog distances
- Record and display last known line run on a gcode file to help with recovery.
- Add delay for erase EEPROM to run before loading the firmware
- Remove grbl setting code
- Add change log
- v.0.0.30: Updated documentation link to BobsCNC webpage
- v.0.0.29: Basic-SENDER initial release
- v1.0.309: OPenBuilds Control Fork
- undefined
- undefined
- undefined

This gives you a quick view of the changes made for each revision.

## FOOTER

### INFORMATION PANEL



The footer at the bottom of the display indicates warning, verifies the com port connection, status of the controller and the line number of the gcode that is running.