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.

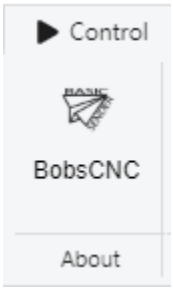
DISCLAIMER

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

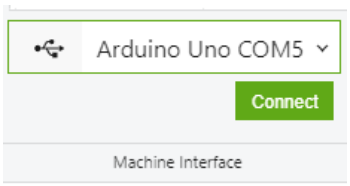
MENU BAR

ABOUT

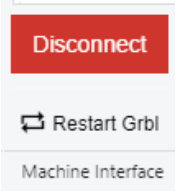


BobsCNC SENDER sends gcode to BobsCNC machines. It uploads BobsCNC firmware. It facilitates troubleshooting set up and runtime. It is based on the open source software created by OpenBuilds.com.

MACHINE INTERFACE



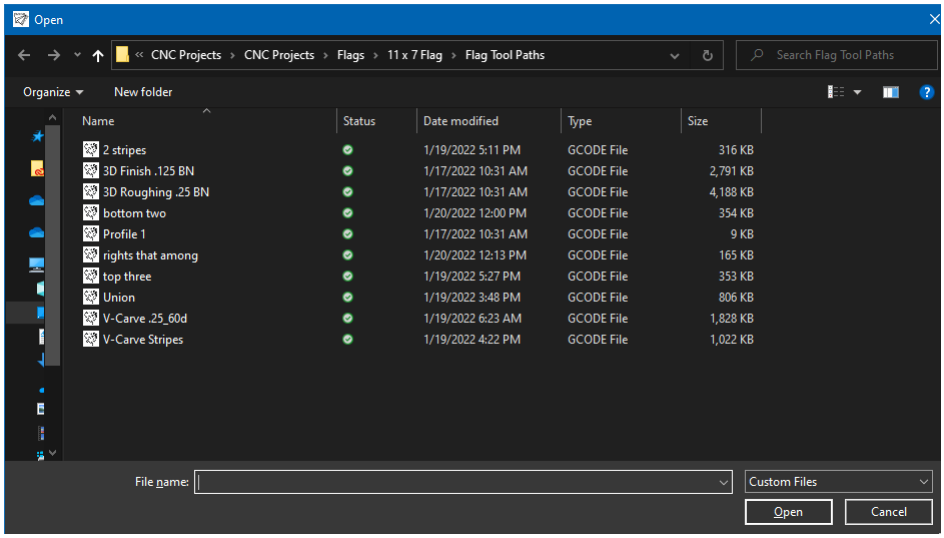
When you connect the controller on your BobsCNC router to your computer with a USB cord, the Status Bar will indicate the type of controller that is being used and the Com Port, which has been designated to link the computer and the controller.



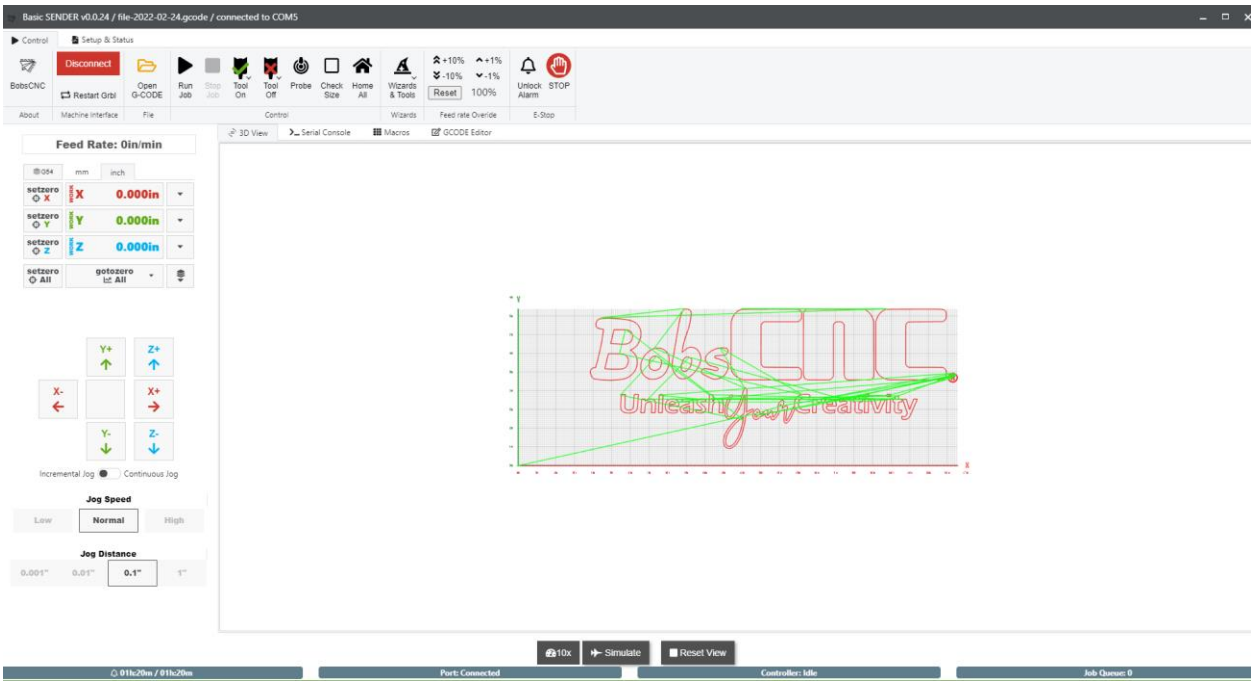
When you CLICK the Connect button, the display will change indicating your computer and controller are connected. I may be different than COM5.

FILE

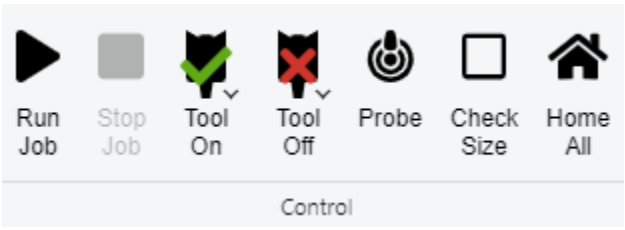
This section of the menu bar contains the Open G-Code icon. CLICK the icon, and you can search for the location where you have saved your gcode files as in the illustration below. Yours will vary.



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

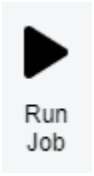


CONTROL



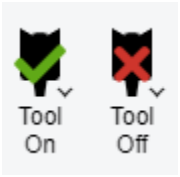
After the gcode is uploaded into Basic SENDER, the Control panel will look this.

RUN/PAUSE/STOP



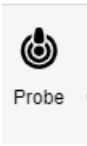
When you CLICK the Run Job button, the gcode will be fed line by line to the controller. The icons will change giving you the option of either Pausing or Stopping your job.

SPINDLE CONTROL

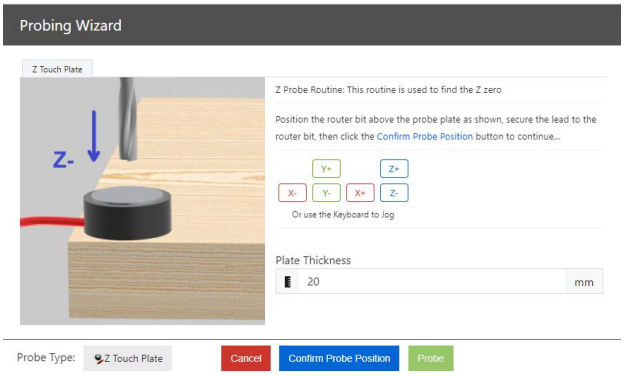


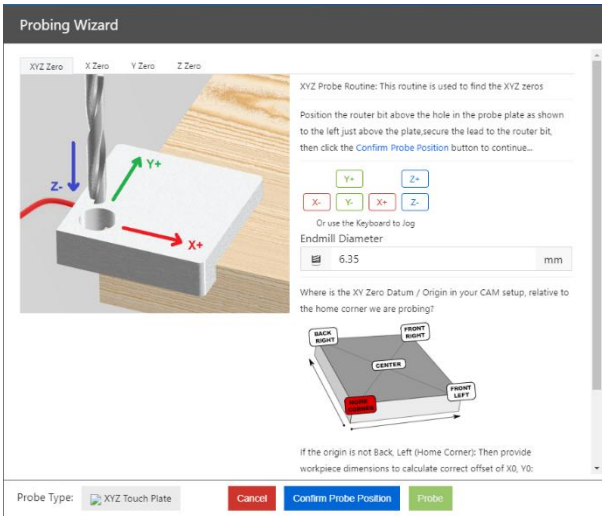
These icons are available for customers who have programed an IOT switch to turn the Spindle on or off. Otherwise, the router can be manually turned on/off using the switch on the router.

PROBE

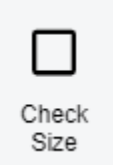


This icon opens the Probing Wizard which allows you to set up a Z Touch Plate and/or a XYZ Touch Place





CHECK SIZE

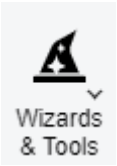


This icon initiates a process to validate that the gcode requirements of the uploaded file do not exceed the X, Y, and Z soft limits to keep from triggering an Alarm 2. It can also be used to diagnose an existing Alarm 2 issue.

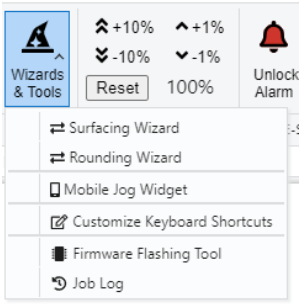
HOME MACHINE

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.

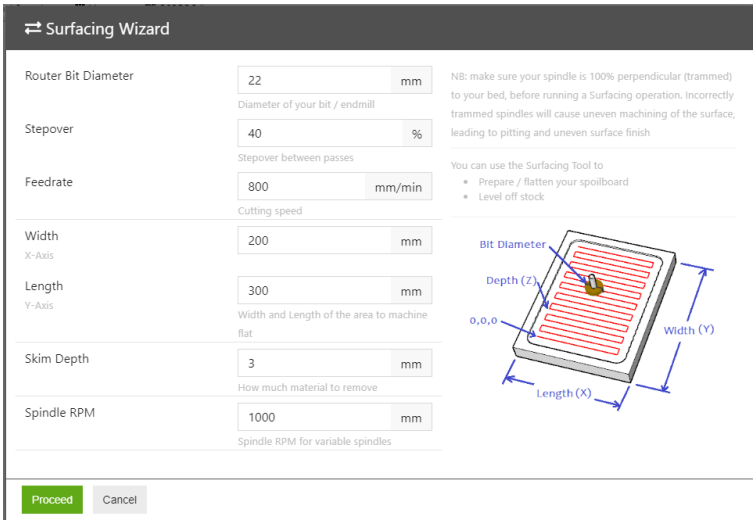
WIZARDS & TOOLS



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.

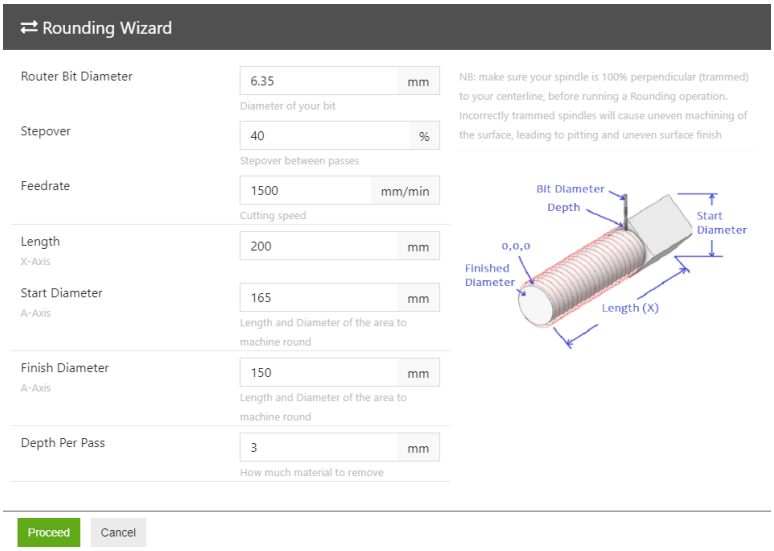
You program in the diameter of the surfacing bit, the percentage of stepover, and the feed rate.

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



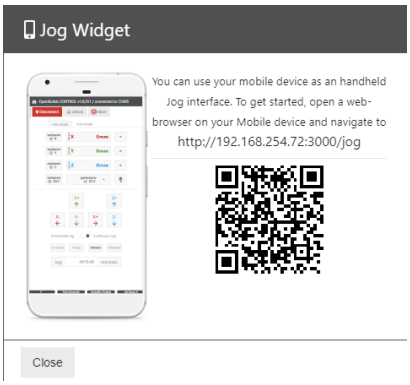
The Rounding Wizard is used with the BobsCNC Revolution Rotary Axis machine. It enables you to transform square stock into round stock.

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.

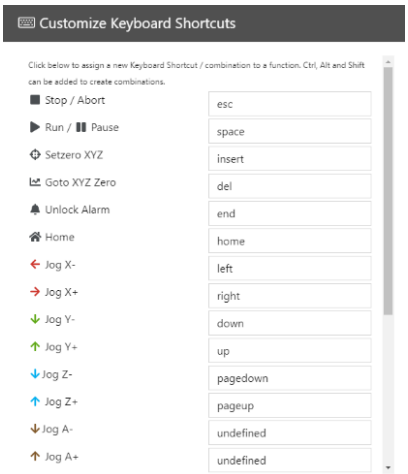
The Depth Per Pass indicates the total amount of material that will be removed in one pass.

MOBILE JOG WIDGET

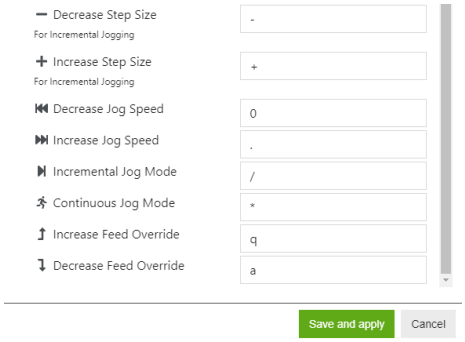


The Mobile Jog Widget is a handy tool that allows you to use your mobile device as a pendant (a handheld Jog Interface).

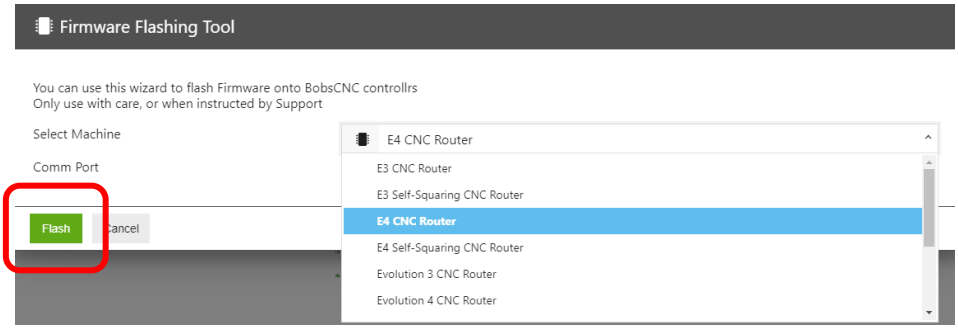
CUSTOM KEYBOARD SHORTCUTS



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

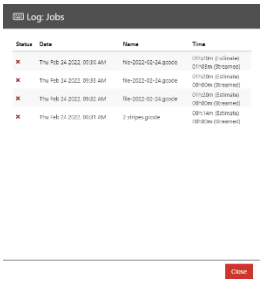


FIRMWARE FLASHING TOOL



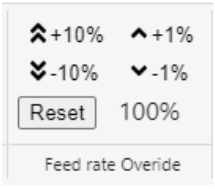
In the event that your firmware values are changed or corrupted, this wizard will automatically reflash the firmware settings on the controller. You simply have to click the drop down menu and highlight the name of BobsCNC machine you are using and CLICK the Flash button (outlined in red above).

JOB LOG



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

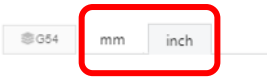
JOG PANEL

FEED RATE DISPLAY

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

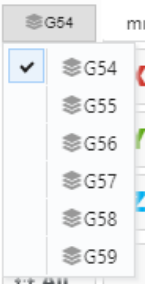
DIGITAL READOUT (DRO)

SET UNIT DISPLAY



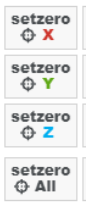
This allows you to tell Gcode Sender if your gcode is in mm or inches

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

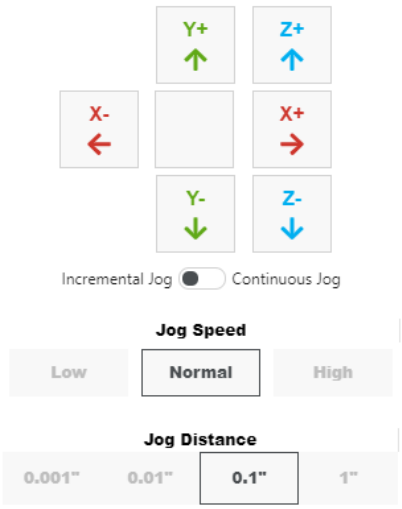
This button will tell the machine to return the spindle to the work zero location

SET PREDEFINED POSITIONS

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

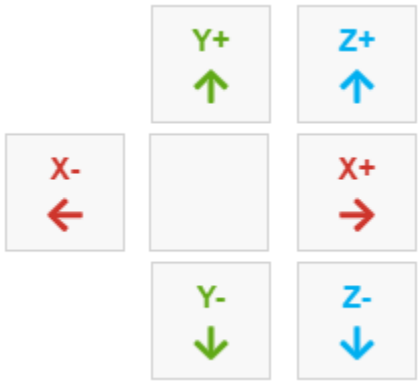
https://linuxcnc.org/docs/2.6/html/gcode/gcode.html#sec:G30-G30_1.

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

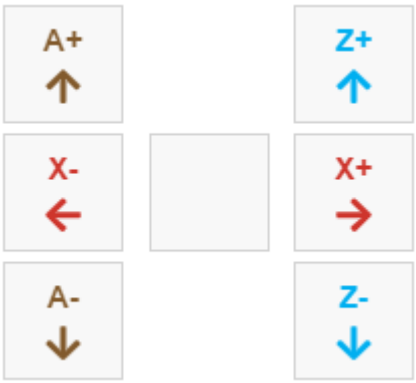


JOG BUTTONS

Standard Configuration (X, Y, Z)



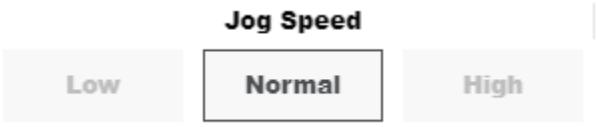
Rotary Configuration (X, A, Z)



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



Low = 12 inches per minute.

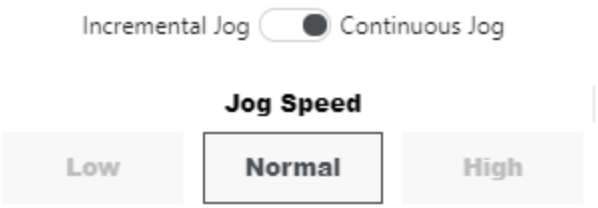
Normal = 197 inches per minute.

High = 394 inches per minute

JOG DISTANCE

There are two settings Incremental Jog or Continuous Jog

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.



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

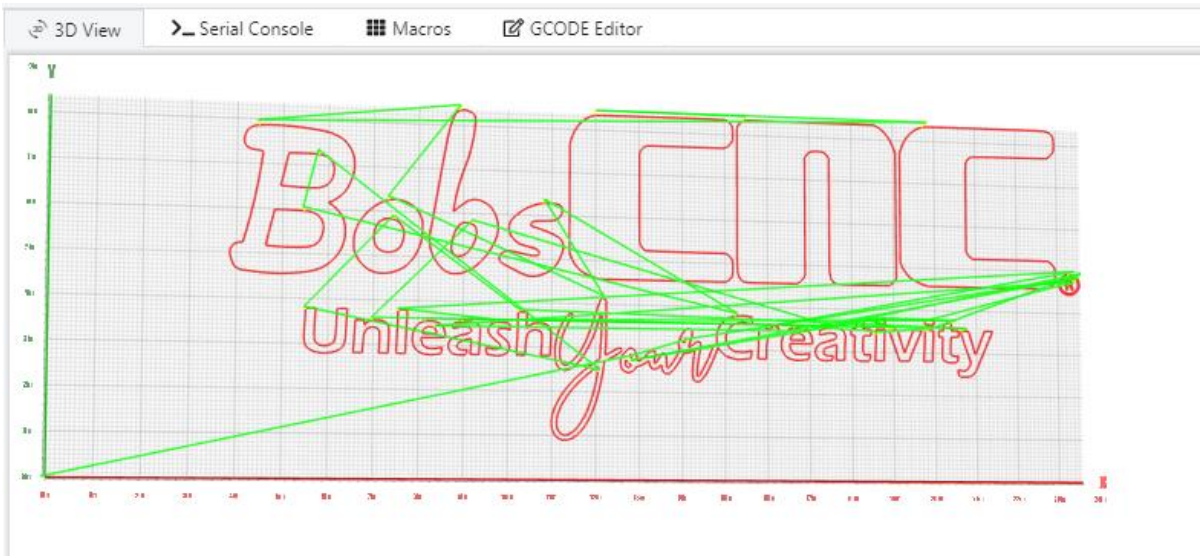
DISPLAY PANEL

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

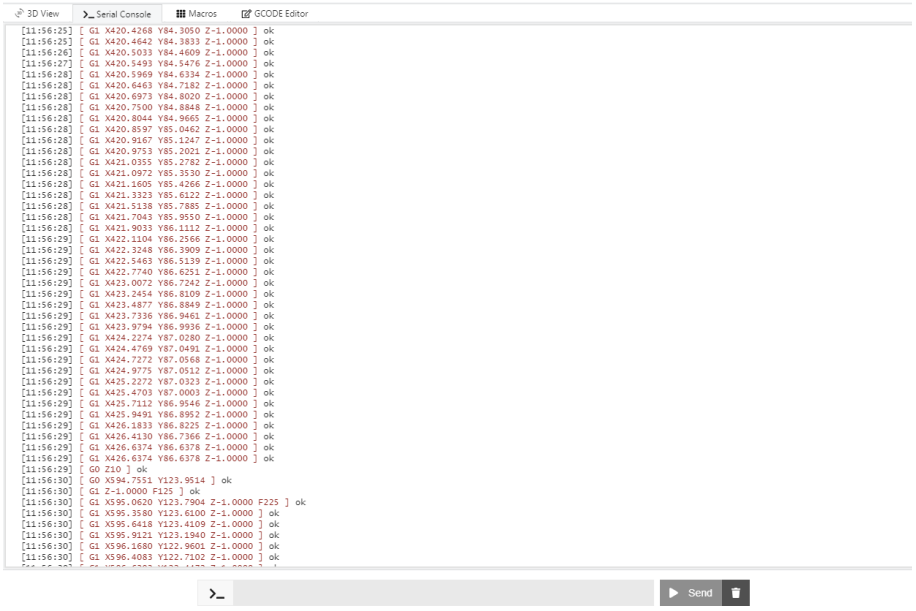
The simulation panel allows you to run your gcode file in virtual reality.

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

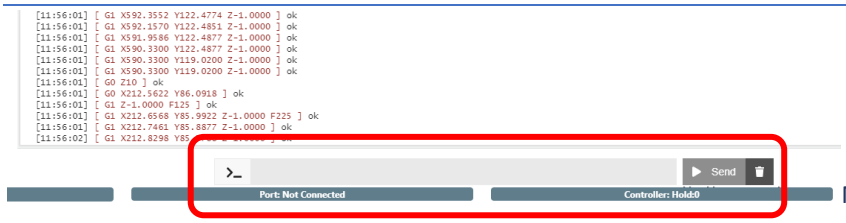
The serial console let you to see the information that is being transmitted via the USB connection

DISPLAY WINDOW



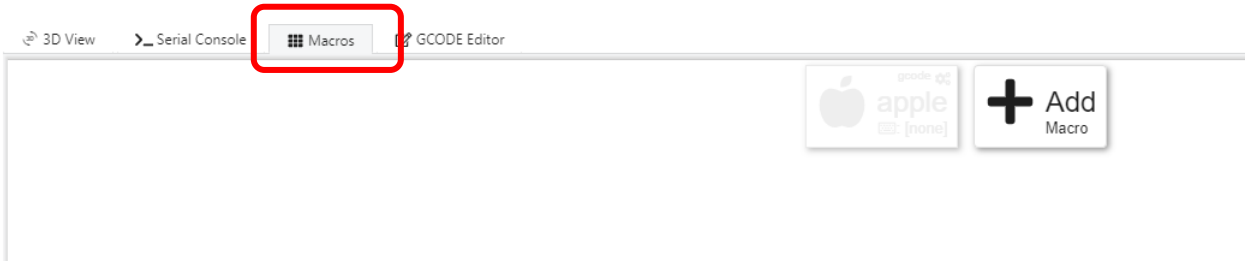
COMMAND INPUT

This where you can insert gcode commands.



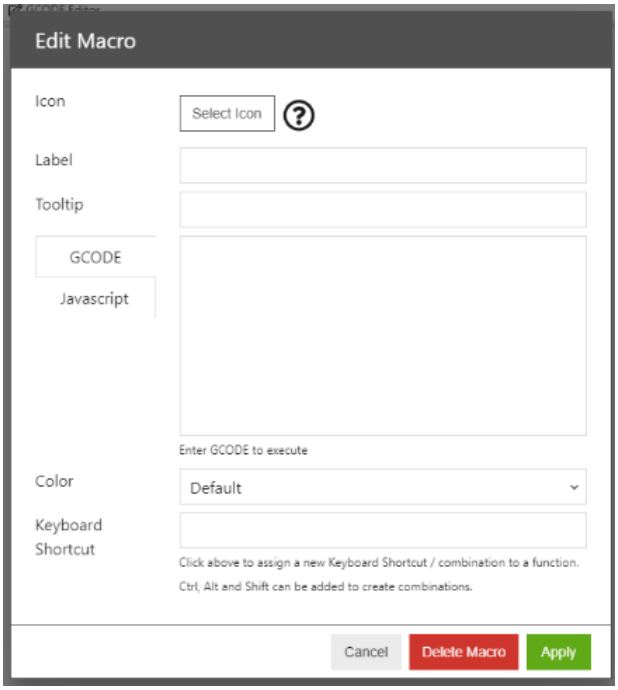
MACROS

MACROS



DISPLAY WINDOW

This window displays all the macros that you have created.



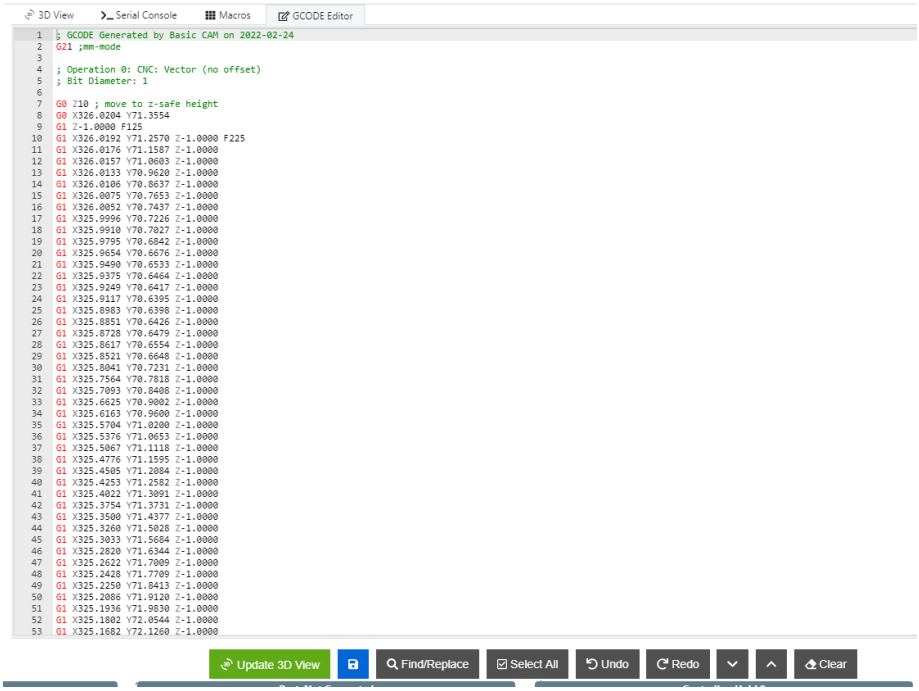
This is the panel that allows to:

- 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.

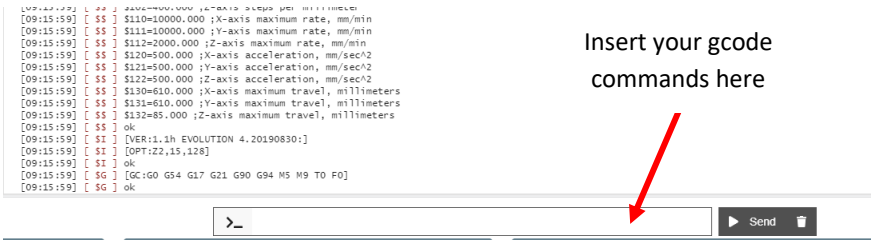
GCODE EDITOR

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

DISPLAY WINDOW

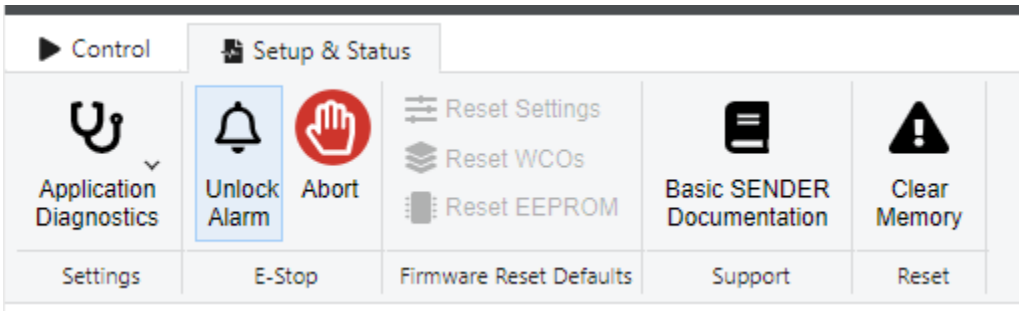


EDIT GCODE PANEL

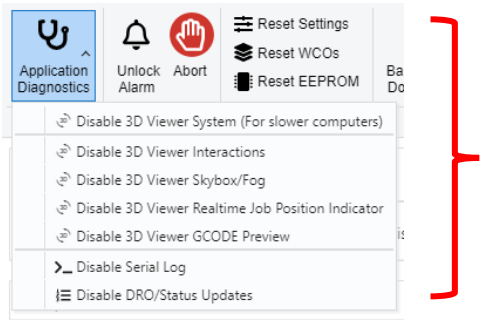


SETUP

MENU BAR

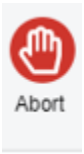


APPLICATION DIAGNOSTICS BUTTON



Select or deselect these functions to tailor your interface experience.

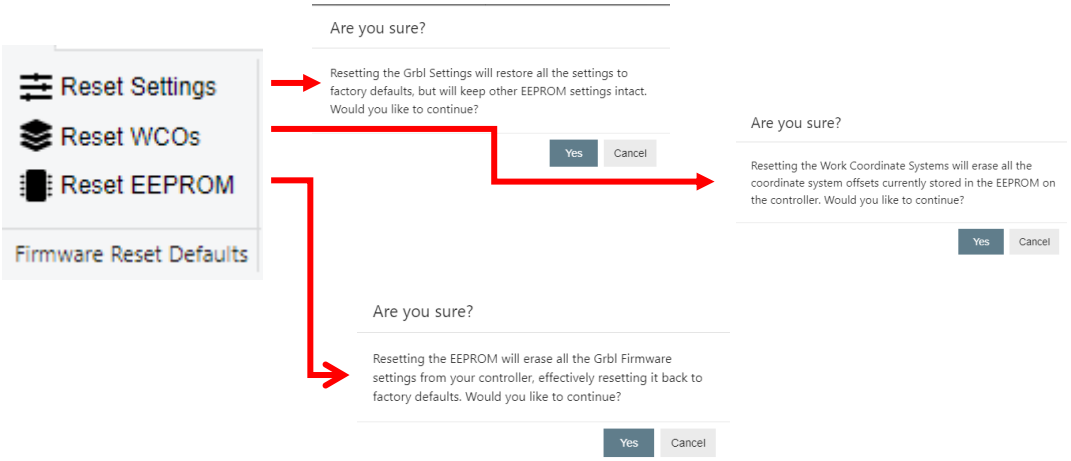
E-STOP



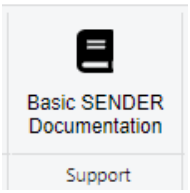
CLICK the E-Stop to stop transmitting the gcode file.

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.

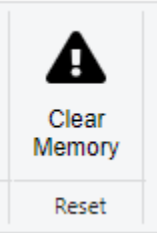


BASIC-SENDER DOCUMENTATION BUTTON



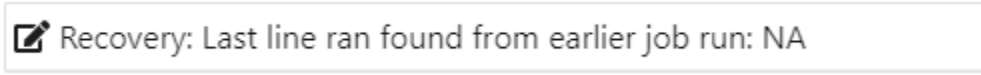
This opens the BobsCNC Documentation page.

CLEAR MEMORY BUTTON



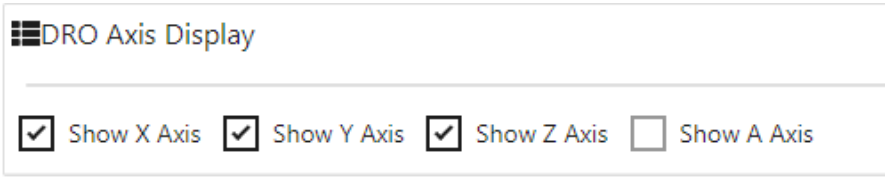
This will set all values ack to the defaults.

RECOVERY



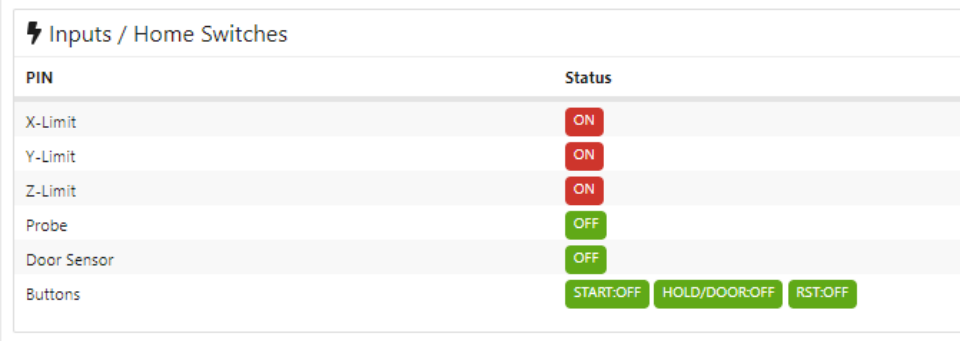
When you run a gcode file this will keep track of the last line ran if the run is stopped or you loose power.

AXIS DISPLAY



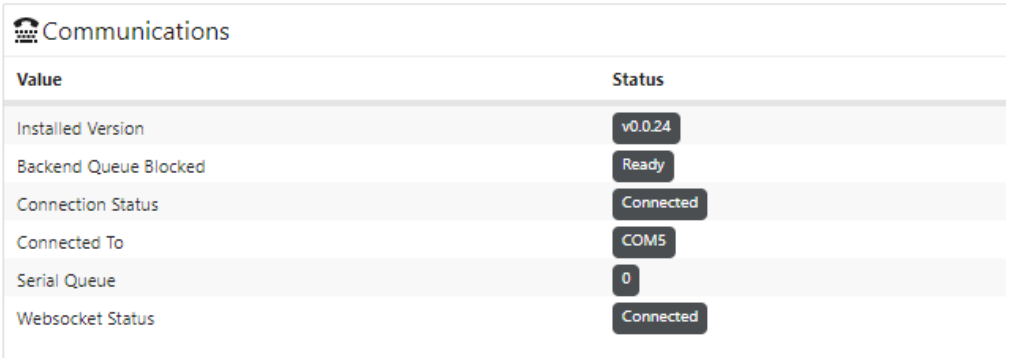
This feature allows you to choose which AXES are displayed. X, Y, Z for the Evolution and KL Series CNC Routers or X, A, Z for the Revolution Rotary Axis CNC Router.

INPUTS & HOME SWITCHES



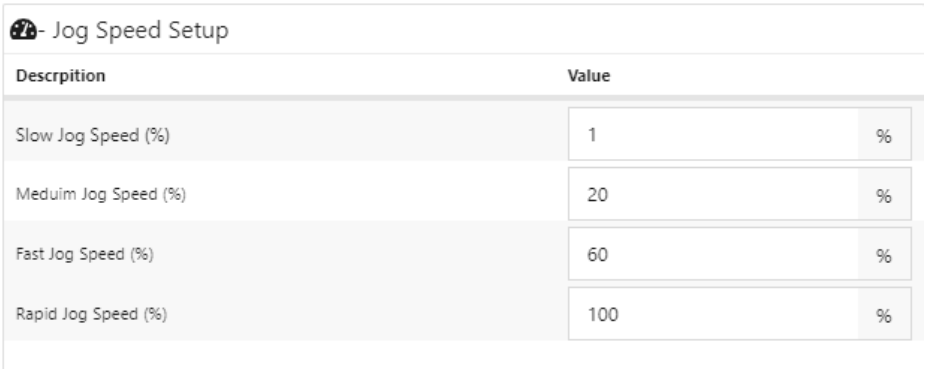
This gives you a quick way to verify the Home switches and other input signals

COMMUNICATIONS



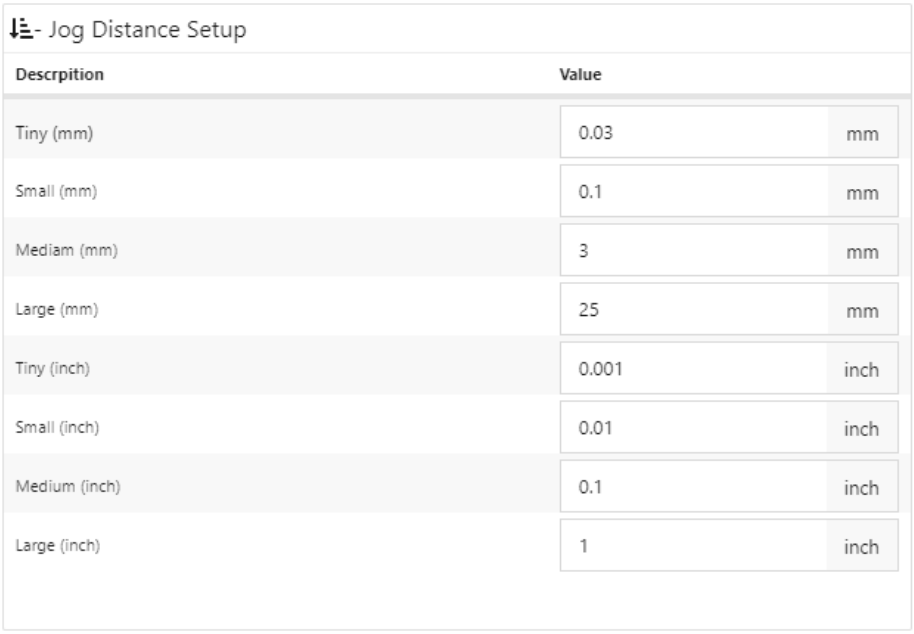
This gives you a quick way to verify potential communications issues with your controller.

JOG SPEED SETUP



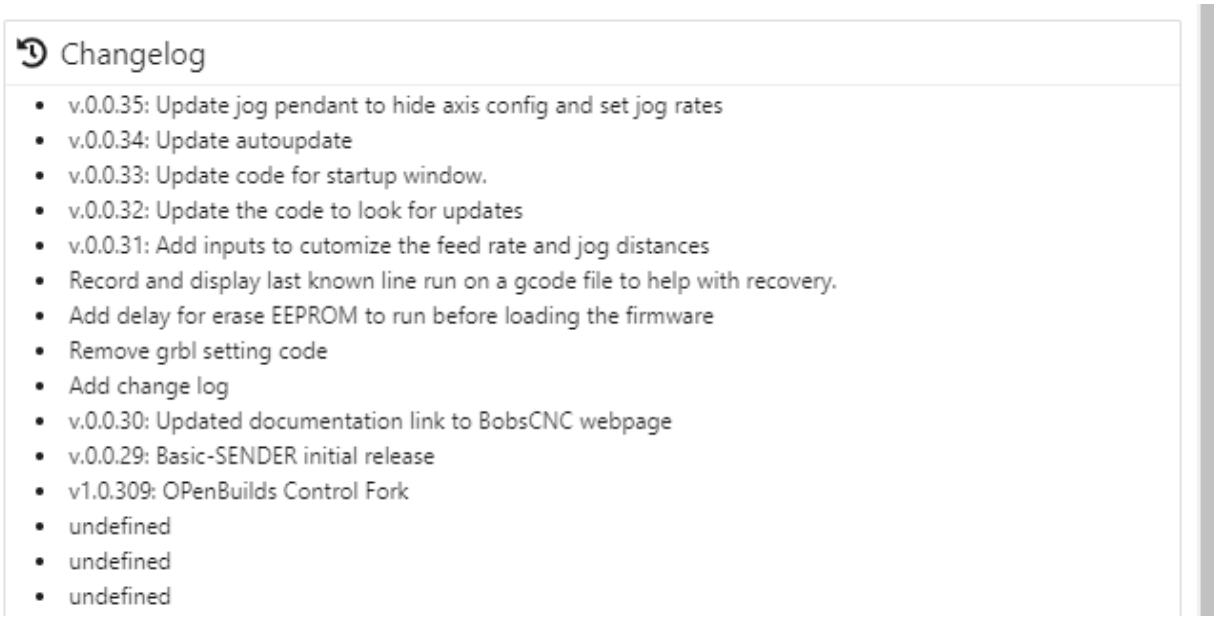
This gives you a quick way to edit your jog feed rates based on the 100% rapid rate set by your controller

JOG DISTANCE SETUP



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

CHANGE LOG



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.