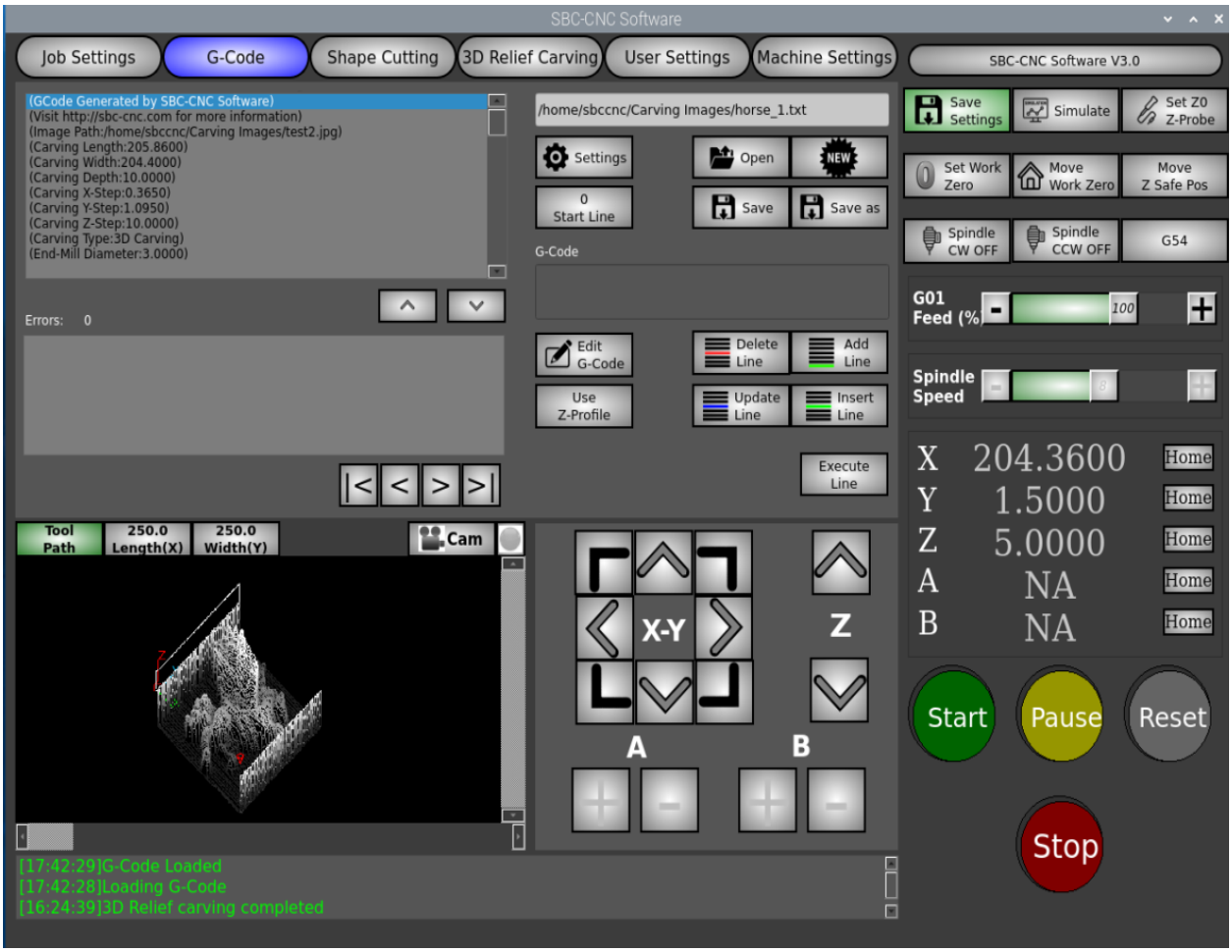
SBC-CNC Software

User Manual V2.0



SBC-CNC Software

Thank you for choosing SBC-CNC software.

This manual provides details about how to install and use the software with your machine. It also helps you to understand different features and configuration of the software. Please read this manual for setting up your machine with SBC-CNC software.

For any queries related to software usage and technical support please reach us at below communication

email: purchase@sbc-cnc.com

Contents

1. Introduction

SBC-CNC stands for “Single Board Computer” CNC software. The software runs on any single board computer, Example: Raspberry Pi 4, and can be used to run any CNC Machine build with stepper/servo (Pulse, Dir) motors.

The software uses on-board GPIO pins of the single board computer to directly control the CNC Machine. You don’t need any other controller. These single board computers use multi-core, high speed ARM processors with large Giga byte RAM. SBC-CNC software utilizes the processing power of these single board computers at near 90% efficiency by deploying multiple threads and task optimization. The software is tested on various platforms and is available for different flavors of Linux distro’s as per your choice.

The software supports additional functions apart from executing G-Code. Shape cutting & 3D relief carving functions makes prototyping and executing your project with ease and fast. With touch-screen friendly software user interface (UI), all functions are kept simple to understand and easy to navigate across. The user settings & machine settings provide vast control of your CNC Machine.

Because SBC-CNC software runs on single board computer, the total development cost of your CNC machine is reduced considerably. The running cost, considering the maximum power consumption of Single board computers, is also reduced.

Along with a dedicated GPIO extension board (sold separately) you can run any Industrial grade CNC Machine without damaging your single board computer.

Special features of SBC-CNC software

- Control using direct SBC (single board computer) GPIO
- G-Code execution
- 2D and 3D Shape Cutting without G-Code. Integrated CAD+CAM Functionalities
- 3D relief carving from Grayscale Image without G-Code
- Custom control of your single board computer (SBC)
- USB camera support for remote Monitoring
- VNC server for remote control

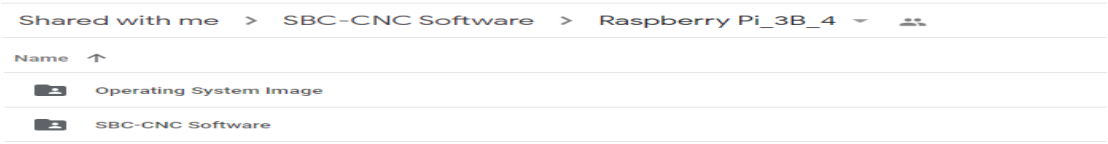
2. Installation

Hardware: Raspberry pi 4
Recommended SD Card Size: 16 GB

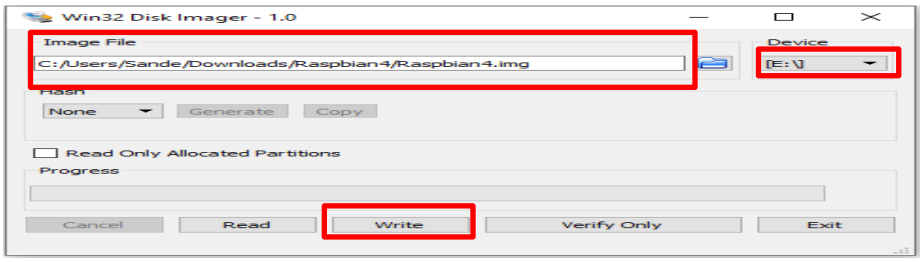
Please visit the following google drive location to download the operating system image for your respective single board computer.

https://drive.google.com/drive/folders/1-O6MgXchQgQKbgEv3WTx6wlPjmRvbcL9?usp=sharing

Download the operating system image along with the latest SBC-CNC software for your Single board computer



Use win32 disk image (or any other image writer application) to write ".img" file into the SD card. Please back-up data from the SD card as it will be completely formatted after this operation.



This will take some time to write. Once the Image is written remove the SD Card from your computer and insert it into your single board computer (SBC).

After booting your single board computer, copy SBC-CNC software to "/home/sbccnc/SBC-CNC Folder" (Refer Page: 27). To run SBC-CNC software double-click icon on your single board computer desktop. Refer Page:22 to learn how to wire your machine.

3. General Features

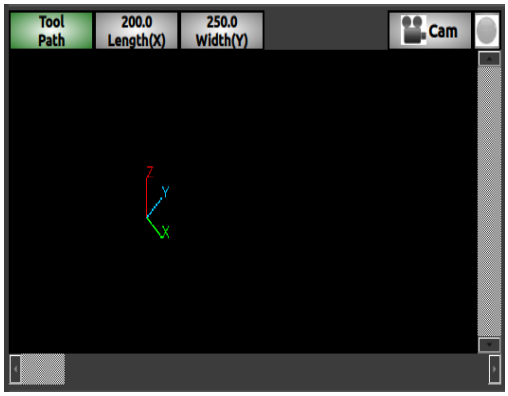
3.3 Tool Path/Camera window

1. Tool Path

This option shows path of the tool only when auto-mode (G-Code/Shape cutting/3D relief carving) is running. During manual operation such as Jog/step/fine motion, no graphics will be drawn. Set the Length and Width options equal to your work-piece size. This will make the graphics appear good. There will be different Gray-shades for different Z values but only when it is negative.

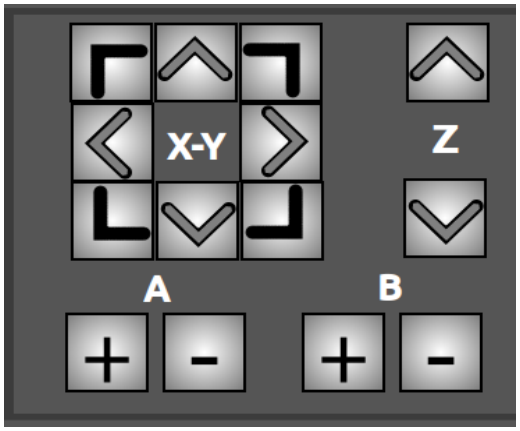
2. Selects Camera

If you have a USB camera connected to your single board computer, the software will automatically detect it. You can visualize your machine operation or record videos using this camera. You can also remotely observe your machine operation when you use VNC viewer from your mobile or a different PC and than connect to the same network on which your single board computer is connected.

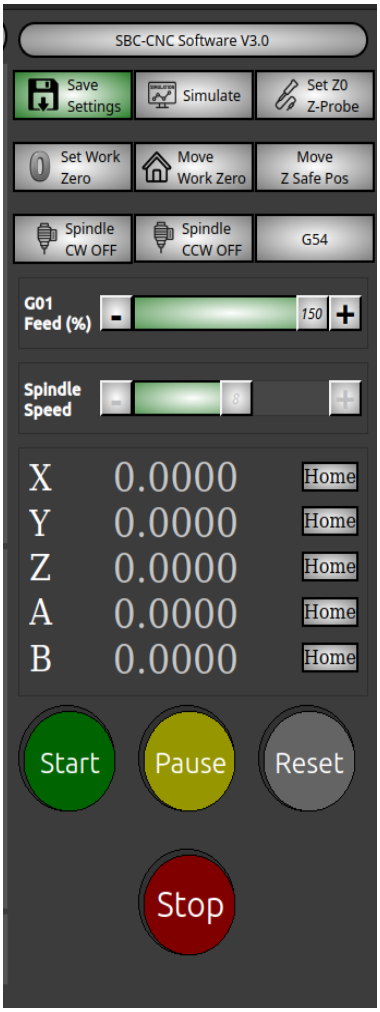


3.2 Manual control window

Manual control options are self-explanatory and can be used to manually move the machine. When the Jog button is released, the machine decelerates.



3.3 Control options



3.3.1 **Simulate** option can be used to test your G-Code. If it is enabled, it will update the positions but will not send pulses to your drive. When it is turned off, X, Y, Z, A & B Positions will be automatically updated to last machine position

3.3.2 **Set Z0 (Z-Probe)** can be used to automatically set the Z-Zero. You can place aluminum foil or any other metal on the work-piece and connect a wire from it to pin number 21 (refer Machine/IO Settings). Connect +3.3V pin on your raspberry pi to your spindle tip using alligator plugs. When "Set Z0" is clicked, Z-Axis moves down and gets in contact with the metal part on the work-piece. This applies 3.3V on pin 21 and SBC-CNC software sets Z –Zero and moves Z-axis 3mm up automatically.

3.3.3 **G01 Feed (%)** can be varied from 10-150. 100 value indicates the feed is equal to user feed value set in the "Üser Settings". 150 value indicates 150% of user feed value.

3.3.4 **Spindle speed** can be varied from 1-15. Each value represents the PWM frequency. The values are determined by the PWM range you selected in Machine/IO Settings.

Example: If PWM range is selected as 100 – 3000Hz, this will be mapped to 1-15; 1 will be 100Hz PWM frequency and 15 will be 3000Hz PWM frequency. You can use frequency to Analog voltage converter to control your spindle speed.

3.3.5 **Stop** will abruptly stop the machine without any deceleration.

3.3.6 To start the program (or) to continue the program after Pause click **Start. Pause** will stop program execution until Start is clicked. To start the program from starting after pause, click **Reset**.

4. G-Code

4.1 Supported commands

G00 (Rapid Move)

G01 (Liner Interpolation)

G02 (Clockwise circular Interpolation)

G03 (Counter-Clockwise circular Interpolation)

G15 (Cartesian co-ordinate selection)

G17 (X-Y Plane selection)

G20 (Unit Selection Inch)

G21 (Unit selection mm)

G28 (Move to Work Home)

G40 (Ignore cutting tool compensation)

G90 (Absolute motion)

G91 (Relative Motion)

M03 (Spindle Clockwise ON)

M04 (Spindle Counter Clockwise ON)

M05 (Spindle Clockwise OFF)

M06 (Manual and Linear Auto tool change)

M07 (Lubricant ON)

M08 (Vacuum ON)

M09 (Lubricant, Vacuum OFF)

M30 (Program end)

M66 (Wait on input)

M80 & M81

M97 (Line jump)

M98 (Program jump)

M99 (Program return)

M200 to M222 (General Purpose Outputs)

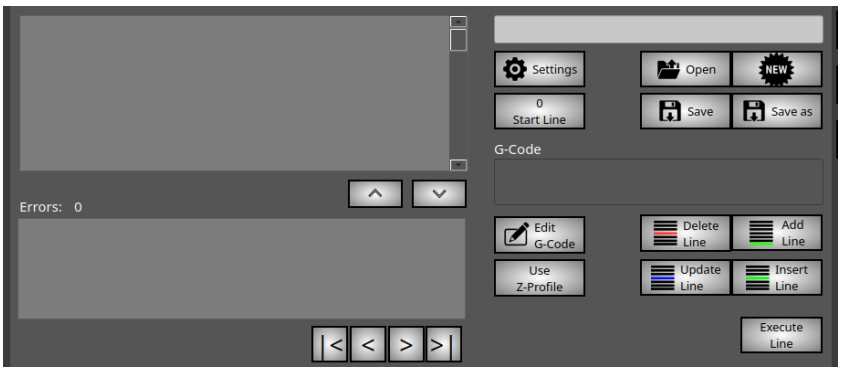
F (Feed adjustment)

S (Spindle speed adjustment)

4.2 Software functions

G-code

G-code window provides functions to load/edit/add and run G-Code. Any change to the G-code is automatically compiled and errors will be shown in the error window. There is no maximum limit on the number of G-Code lines. All the lines will be loaded at the same time.



G-Code Line editor

The user can add G-code line and press “ENTER” key on the keyboard to add it as last line in the G-Code file.

Error display window

Error display window display any error in the G-Code or if there are commands which are not supported by SBC-CNC software. User can also edit and update the G-Code line.

Execute single line

This option will enable user to execute one line at a time. The index will automatically move to the next line after executing the current G-Code line. Repeatedly pressing this button will allow user to run manually test G-Code online at a time. User can run any line from the G-Code file, simply by selecting that specific line in the G-Code viewer.

Settings

Use Program Feed makes the feed to change according to the G-Code program file. Disabling this would make the motors run at a constant feed

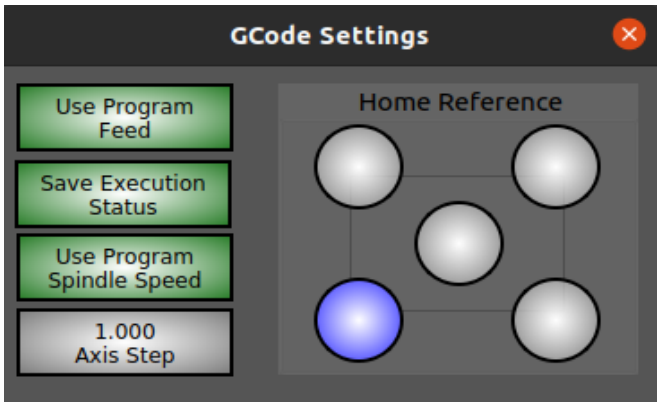
Save Execution status will save the current execution line. The saving happens in a cycle after every 10 lines gets executed. In case you lose power to your single board computer, you can use this option to continue your work without starting fresh again.

When you reload the same G-Code file, SBC-CNC automatically jumps to the last execution line

Use Program Spindle speed makes the spindle speed to change according to the G-Code program file. Disabling this would make the spindle to run at a constant speed.

Axis step provides minimum step used by axis while calculating steps for Circular interpolation. Software automatically decides which of the axis will use this step.

Home Reference is used to select the origin in the “Tool Path” window mentioned in section 3.1. This is used only to enhance the visualization in tool path window and has no importance in executing G-Code. Based on origin selected during G-Code creation, select the option here for better visualization in tool path window.



5. Shape Cutting

Shape Cutting is an added functionality of SBC-CNC software to cut 2D and 3D shapes with ease without and G-Code. This is quite useful for newbies. It combines CAD+CAM in a single utility.

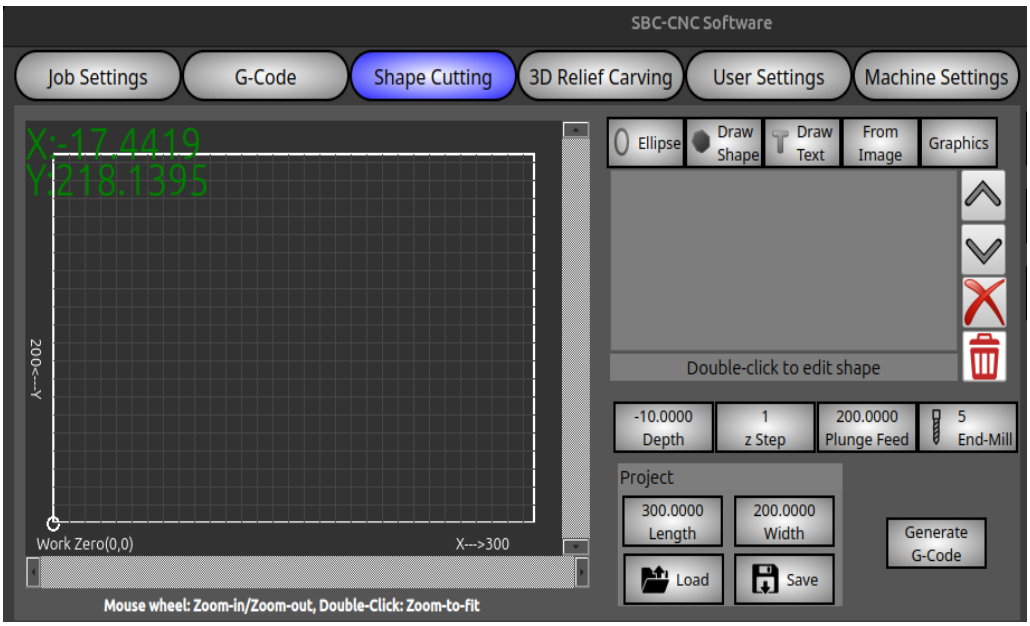
You have multiple options to create a shape: Manually **Draw Shapes** with simple mouse clicks or let SBC-CNC software generate shapes **from images**.

You can directly cut/carve Text using **Draw Text** Option

Each shape can have its own depth. Creating shapes enclosed within each other and of different depths will create 3D shapes. For every shape added, you have option to either cut only the periphery, or cut it as a **Pocket**.

User can change the shape position or orientation at any later point of time after the shape is created. Double-click to edit the shape and each point/vertex can be modified.

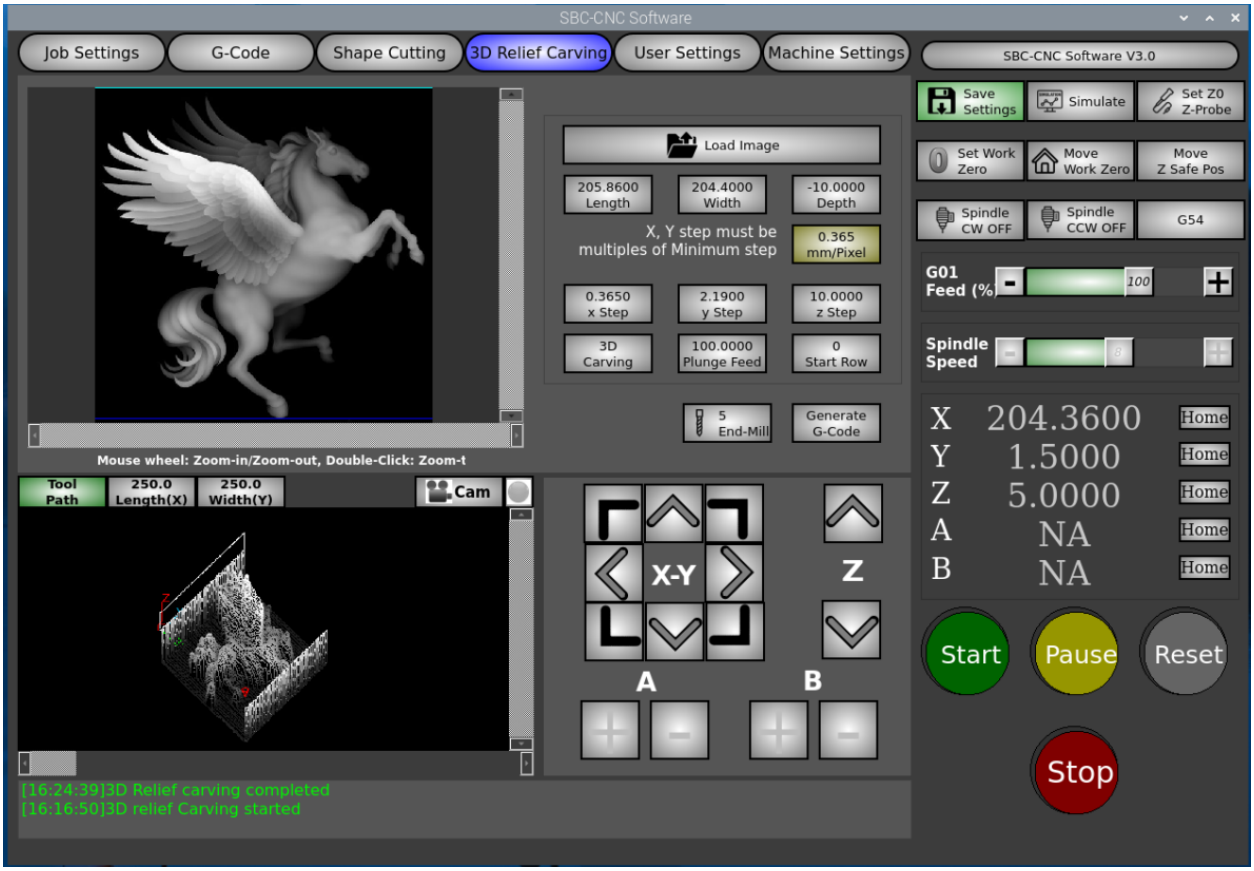
Shape cutting only runs on the X-Y positive plane. You work home must be always be set at the bottom left of X-Y Plane.



6. 3D Relief Carving

3D relief carving function can carve any gray-scale images into beautiful 3D reliefs. The function is quite simple to use. You can also do 2.5D carving using this function (in 2.5D carving X,Y and Z will not move at the same time)

3D relief carving function runs on the X-Y positive plane. Your work home must be always be set at the bottom left of X-Y Plane



7. User Settings

User settings are used to control the machine by the user. Most of the controls are self-explanatory. **Save Settings** must be pressed to retain the settings when the application is loaded again

All the inputs to the software are in mm, rpm & seconds: Distance: mm, Feed: mm/sec, Acceleration: mm/sec^{2,}, Spindle speed: RPM. However, in G-Code, user can use G20 command to change the units from mm to inch. When G20 is mentioned in G-Code, the distance and feed are measured in inch & inch/min respectively.

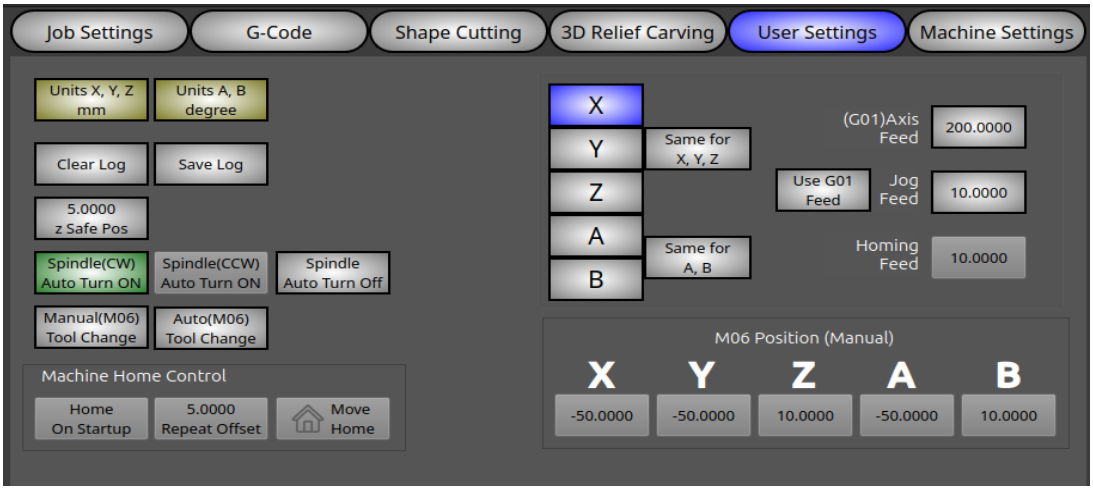
Use G01Feed will set the Jog Feed to change with G01 Feed (%)

Axis Feed: This is the user selected axis feed. This is used by, G01, Shape cutting and 3D relief carving. The maximum value of axis feed is equal to 100% of **Max Feed (G00)** set in Machine settings. For example, If the maximum feed is set to 150 mm/sec, the user Axis Feed can be set maximum to 100mm/sec only

Jog Feed: This is the feed used for Jogging the axis. The axis will decelerate when jogging is stopped as per the acceleration value provided in the machine settings. Keep this value less to avoid jerks.

Machine Home Control
This option is enabled only when home sensors are used and enabled in **Machine Settings**
Repeat Offset: When Machine Home is clicked, all axis will move with jog feed towards home sensors. After reaching home sensors, all axis will move in opposite direction to home sensors by **Repeat offset**. After this, the axis will move again towards home sensors with **Homing Feed**. This procedure is used to reach home fast and with high accuracy. Keep the Jog feed minimal to achieve high repeatability in homing position.

Z Safe Pos: If home sensors are enabled, this is the safe position of Z-Axis from Machine Zero. If home sensors are not enabled, this is the position of Z-Axis from work zero.

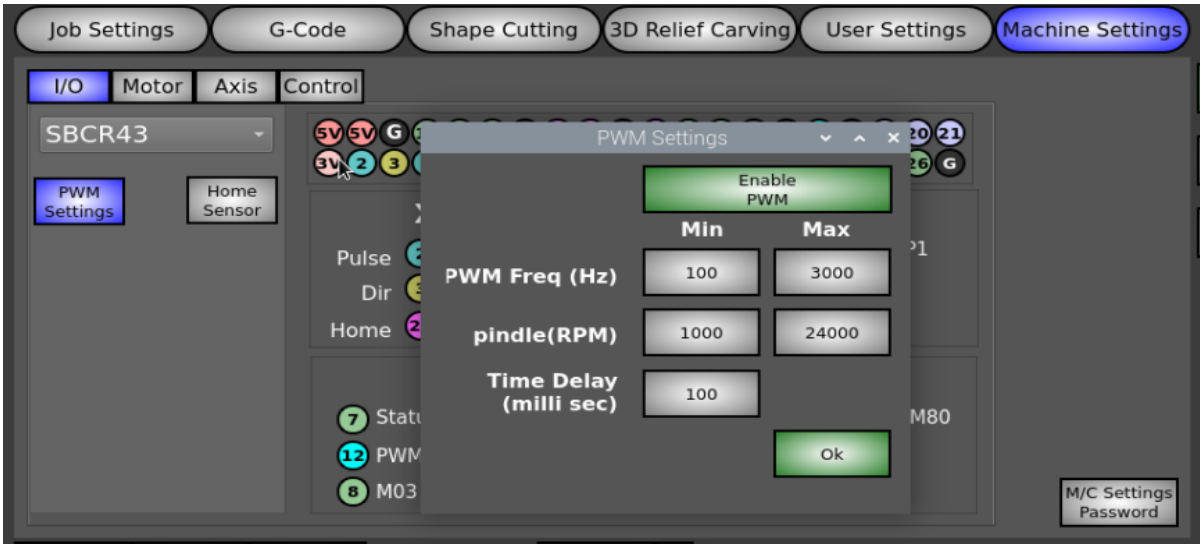


8. Machine Settings

SBCR43 is for 3-Axis machine **& SBCR45** is for 5-Axis machine

Raspberry GPIO's can provide only 3.3V. Please be careful and avoid giving input voltages above 3.3V to avoid damage to your raspberry pi

PWM Settings: This window allows you to set Spindle speed control. You can select **PWM Freq** range and the respective **Spindle RPM**. **PWM Freq** and **Spindle RPM** will be mapped 1 to 1 with 15 steps. **Time delay** is the wait time after spindle speed is changed before the next motion command is executed.



Motor Settings

There are two types of Feed curves supported: Trapezoidal and S-Curve. In Trapezoidal curve, the acceleration is a step curve and in S type curve, the acceleration is a Trapezoidal curve. The S-Curve smooths the motion characteristics. For the S-Curve, user has to provide the “Rate of Acceleration” in mm/sec^3 to define the slope of the Trapezoidal acceleration curve.

Sampling Time

It is the frequency at which you select how fast the feed increases from one value to the next. Keeping this value very less will make the curve very smooth as the feed will change very slow and increasing this value will reduce the smoothness as the Feed will change faster.

Move/Pulse

How much linear distance the axis moves per 1 pulse

Max Feed

The maximum rated Feed of the machine **G00**

Acc

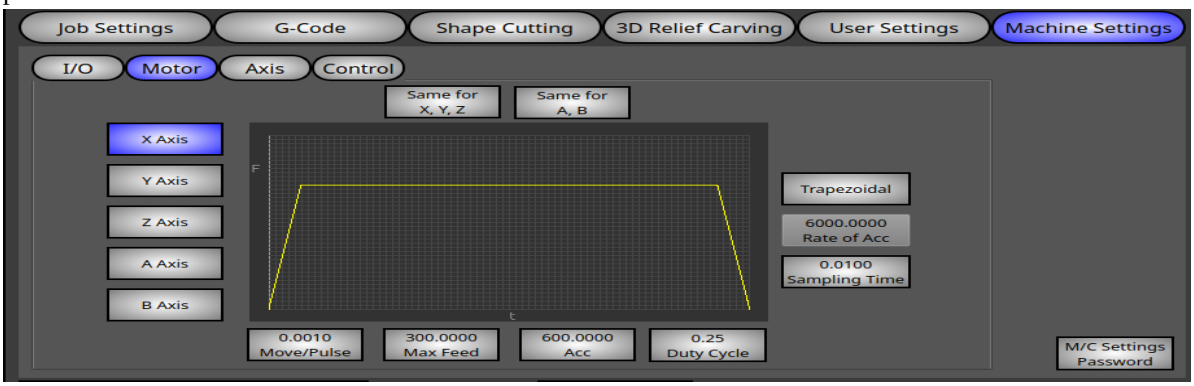
Acceleration of the machine

Duty Cycle:

The ratio of Pulse On-Time/Total pulse time applied to motor driver

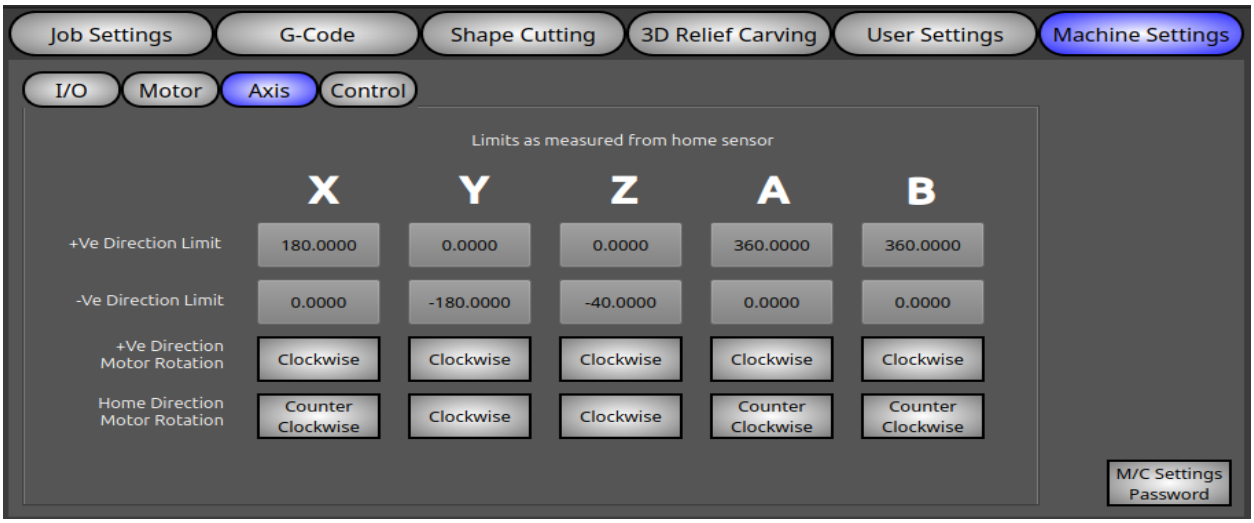
M/C Settings Password

This password is used to merely control accidental changes to Machine settings. It doesn’t provide any security protection for the manufacture settings. Deleting the SBC-CNC.dat file will reset the password



Axis

These settings will control the limits and Directions of the CNC Machine. Limits will be available only if **Home sensors** are used.



Control

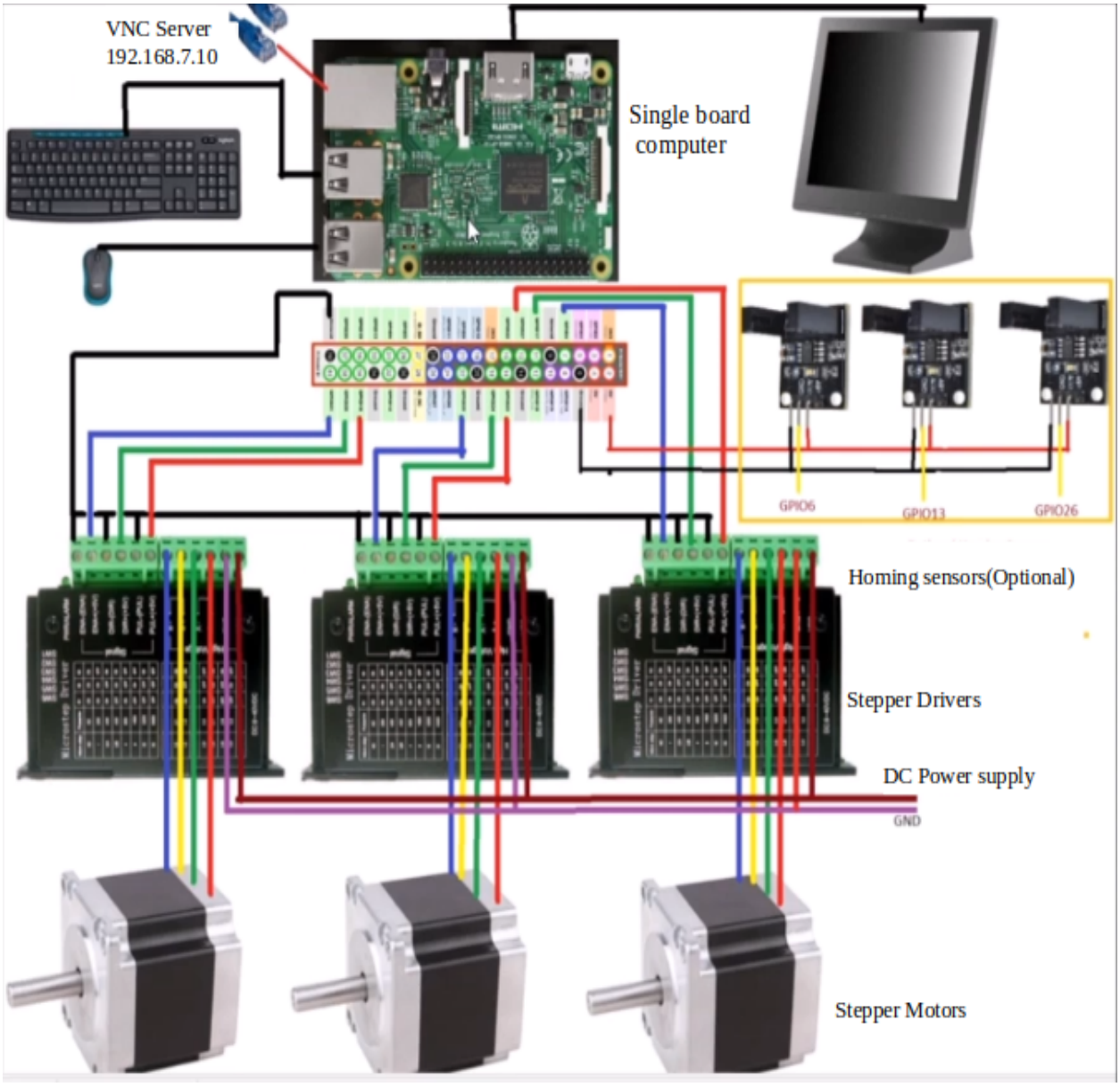
Using a numeric keypad you can control your entire CNC Machine without connecting to any monitor or mouse. Use a wireless numeric keypad and enable **Keyboard Control**. You can printout the stickers from google drive location mentioned in https://sbc-cnc.com

Each key has multiple uses. Please read the key assignment in the software.

9. Wiring

Wiring should as per the pin assignments as mentioned in **IO Section.** Below image is for representation purpose only.

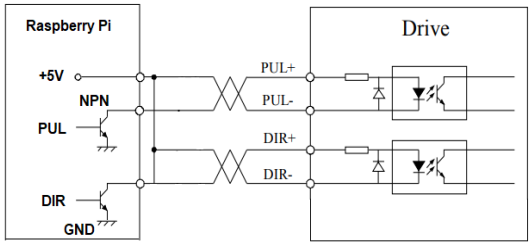
PUL+ and DIR+ must connect to the pins as mentioned in the IO Settings
PUL- and DIR- must be connected to any of the ground pins on the raspberry pi IO.



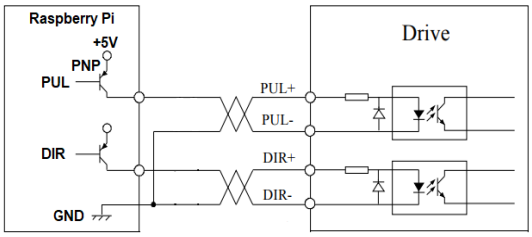
Missing Steps because of Low Voltage

Most of the stepper drivers work with 3.3V PUL and DIR Inputs. However, based on your wiring length and cables used, some stepper drivers can lose steps because of low voltage. If you are observing your stepper motors are losing steps, you need to use additional HW and change the wiring as shown below to boost your PUL and DIR voltage levels

You can use either NPN Transistor OR PNP Transistor configuration



Connections to open-collector signal (common-anode



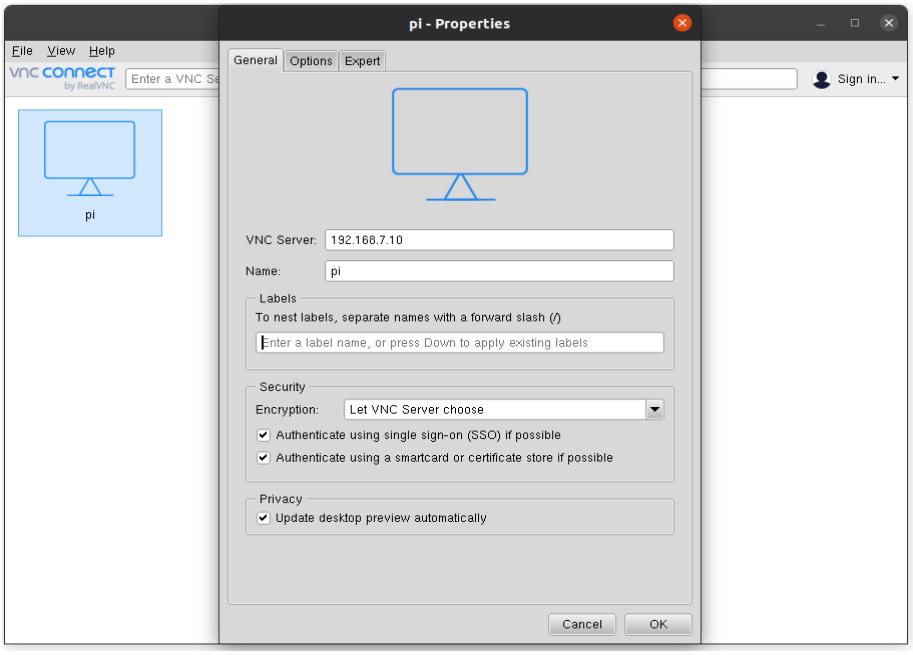
Connection to PNP signal (common-cathode)

10. Remote Control using VNC Viewer

Install VNC Viewer in your PC (Laptop, desktop, android/mac mobile, tab)

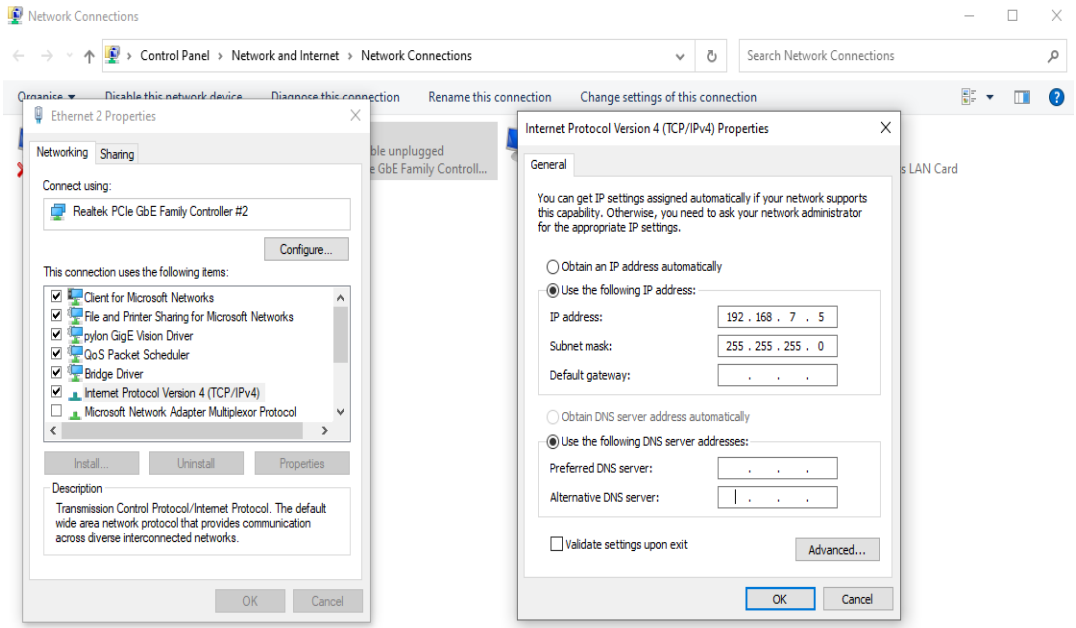
VNC Viewer : https://www.realvnc.com/en/connect/download/viewer/

After installation, you should see the below screen
Click on File and select new connection. Enter details as shown below and press ok

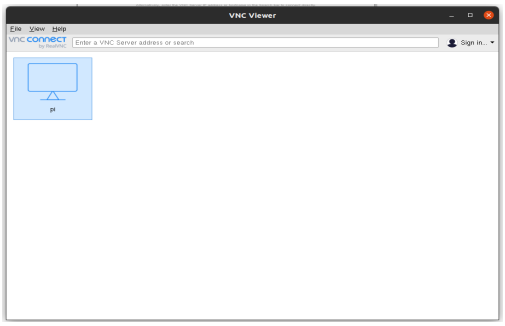


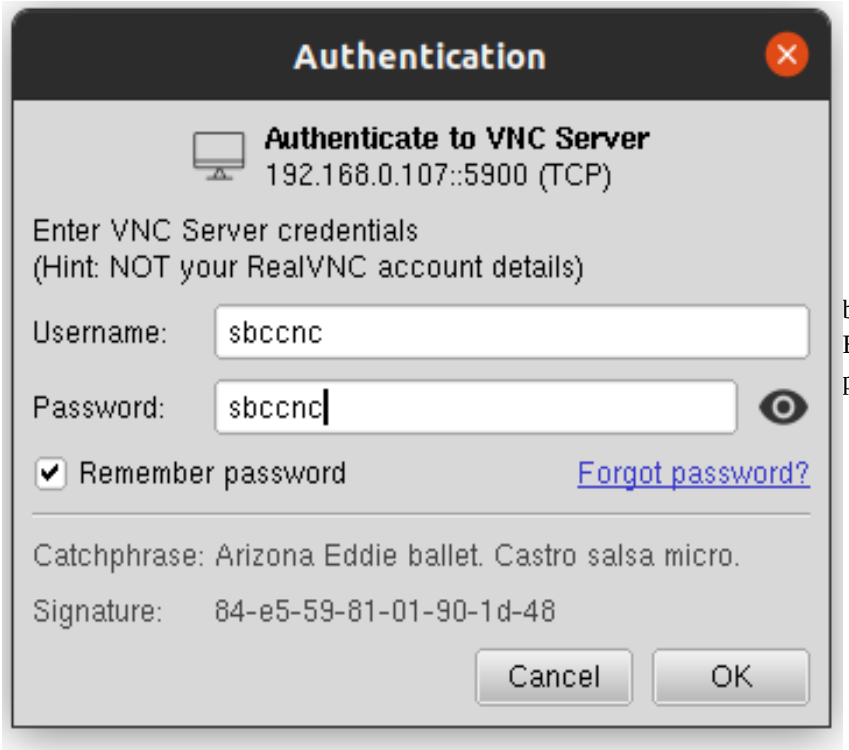
Connect a Network/Lan cable between your pc network port and single board computer on which SBC-CNC is installed.

Configure your pc network port exactly as shown below and press ok.



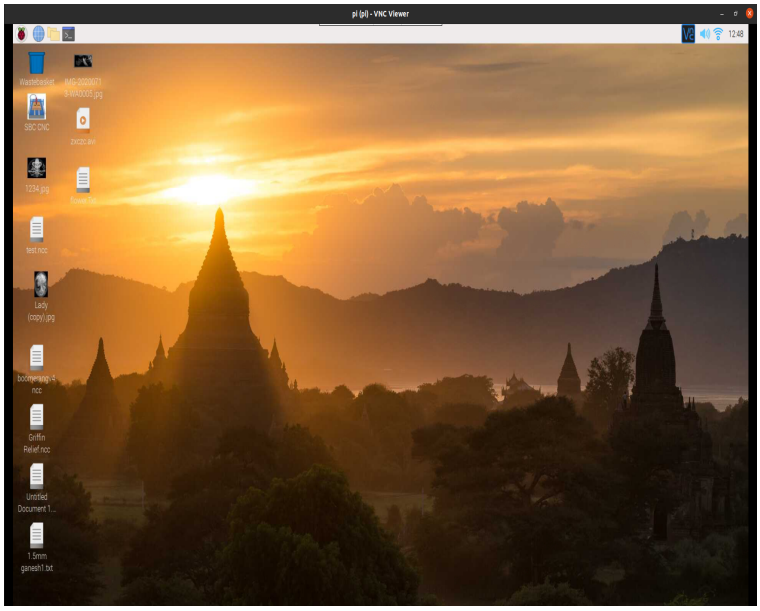
Double click on the icon in the VNC Viewer





For the first time below screen will appear. Enter details as shown and press OK.

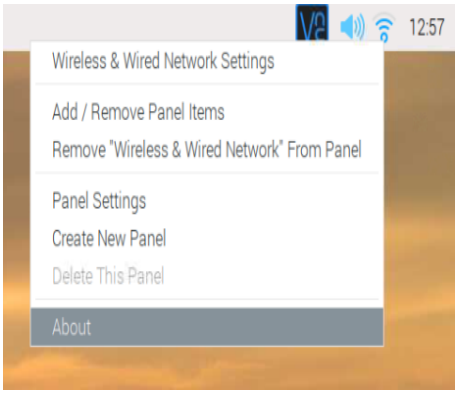
You must now be able to control your single board computer remotely



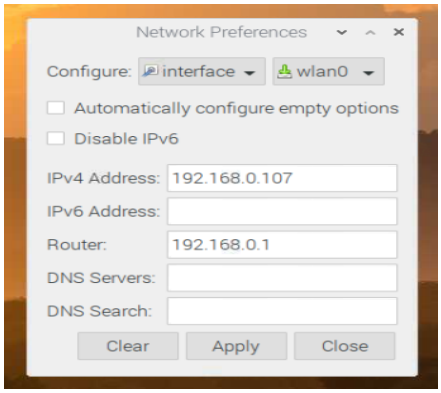
You can repeat the same settings to connect on a wifi network. But for the first time, you need to connect and configure the IP assigned to your single board computer by your wifi router. You might want to assign a static IP to your SBC, on your wifi network, so that the IP address doesn't change every time when there is a power recycle. All the other steps remains the same, except you don't have to configure your PC's network port manually as your router will assign your IP on the same subnet.

To set a static IP to your SBC follow procedure as shown below. Below procedure is only applicable for raspberry pi but will be similar for other SBC's

For Raspberry pi, right-click with your mouse on the Wifi-Icon as show below anc click on "Wireless & Wired Network Settings" option



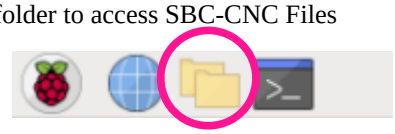
Enter details similar to what is shown below. In this example, the router is assigning addresses in the range 192.168.0.2 to 192.168.0.250. You can manually select one of them which is not assigned by your router to any other device. Click apply and reboot your device.



Now the user can connect to the raspberry pi from any device using the new assigned IP on the wifi network.

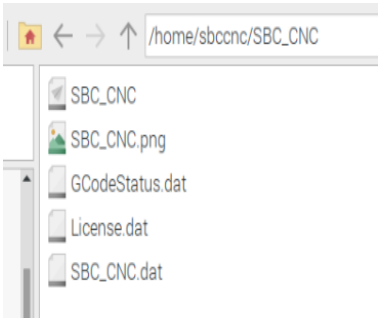
11. Folder structure

By default all single board/linux environments will have the home folder. Please go through the home folder to access SBC-CNC Files



Double-click on SBC_CNC Folder in the home directory.

Following files must be present in the SBC_CNC folder.



SBC_CNC is the application file

License.dat contains your license file for the application. When you would want to purchase the license for the software, you need to send the serial number of your single board computer to "purchase@sbc-cnc.com". To find your serial number just follow the installation procedure as mentioned in this document and open the SBC_CNC software . This will show a pop-up message with the serial number of your single board computer.

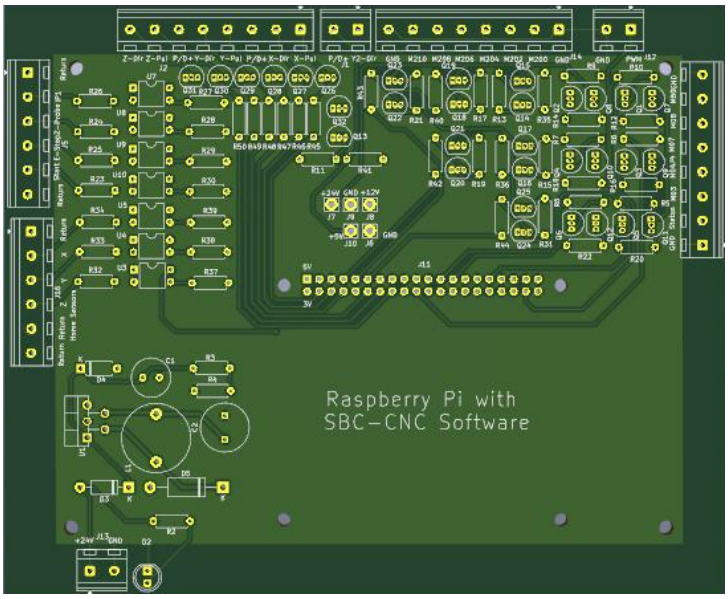
GcodeStatus.dat file is used to save the execution status of your G-Code file.

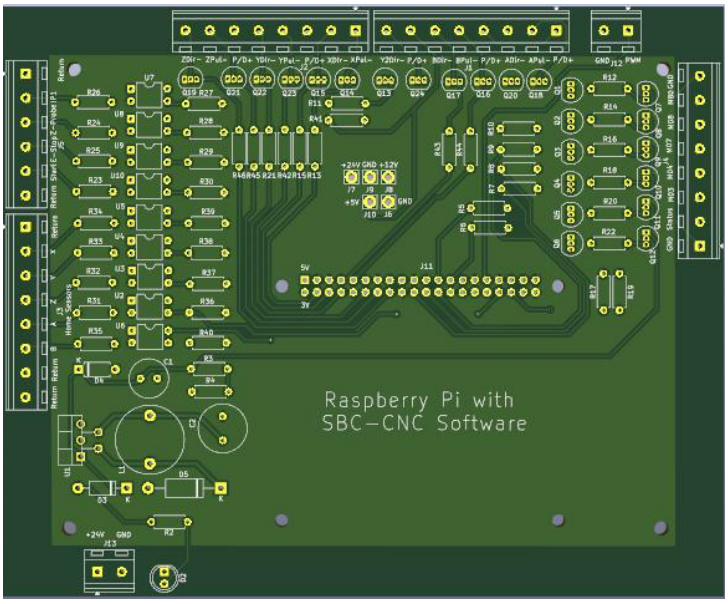
SBC_CNC.dat contains all the settings of your software. This file will be automatically created. In case there is settings issue, just delete this file and the software will reset to default state.

12. SBC-CNC Controllers

3-Axis & 5-Axis CNC controllers. You can build your own CNC controller products with SBC-CNC Software. All the Gerber files to manufacture these PCB's are available at https://sbc-cnc.com

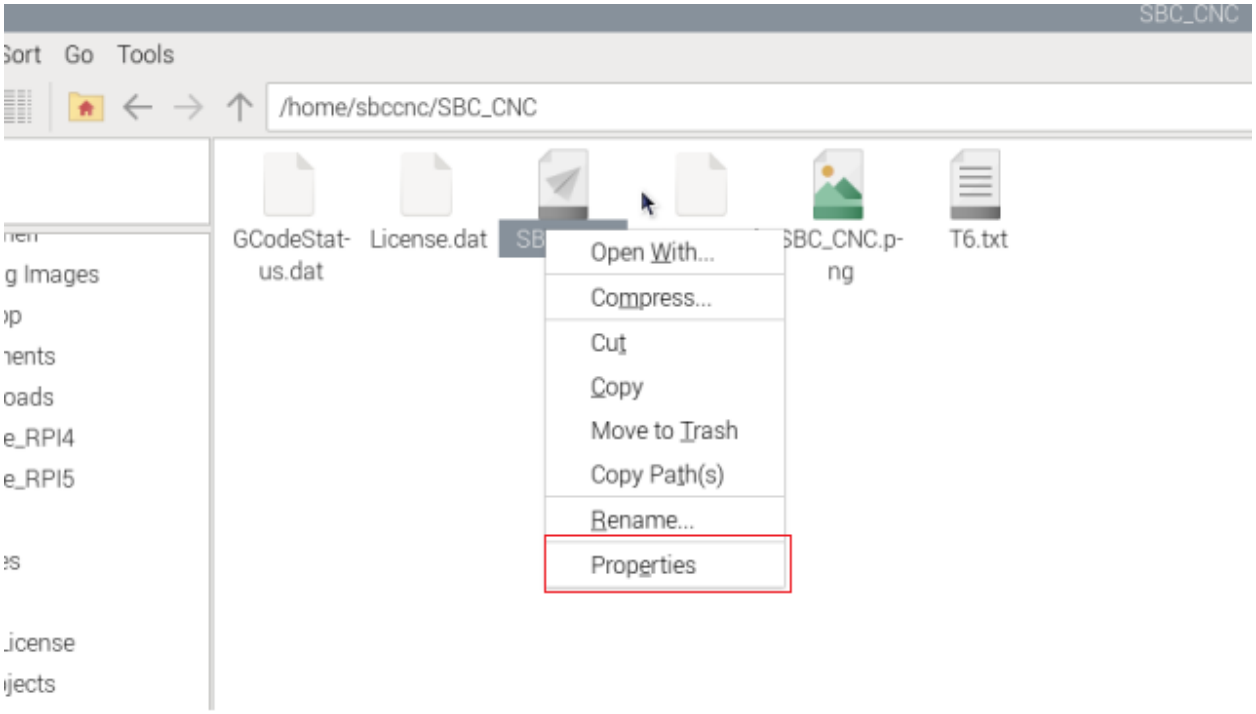
You can download these Gerber files for free and manufacture these PCB's locally or you can buy these controllers from us.

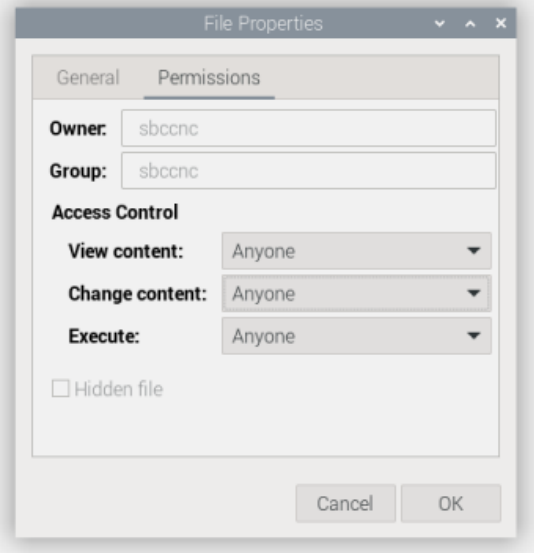




13. SBC-CNC Software Opening issue

If you are unable to open SBC-CNC software after copying new application file into your home folder, please set the permissions as shown below to resolve the issue.





14. Appendix

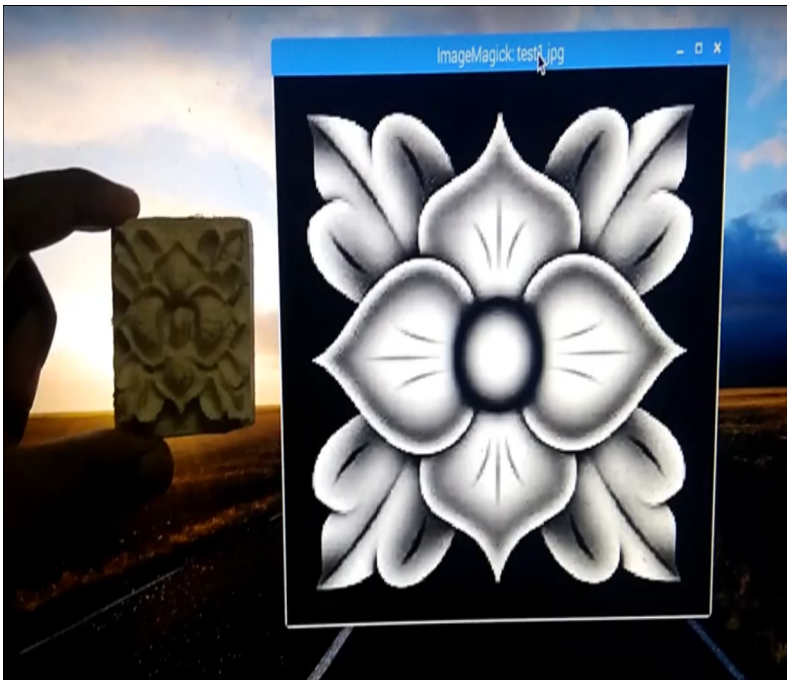
Bhart-Mech Industrial CNC Machine, Mumbai, and SBC-CNC software

3D Relief carving using G-Code

Hobby CNC with SBC-CNC Software

Ganesh 3D relief carving

3D Flower Carving



2D shape drawing on Hobby CNC