GRBL Control Software Introduction
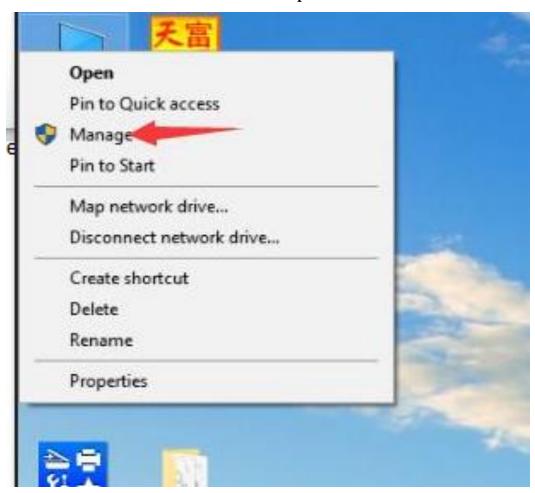
1. Assembly Please refer to 《Assembly instructions》 to assemble the machine。

   ● **Debugging**

2. Install the driver CH340SER

CH340SER

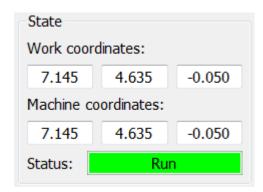3. How to find the correct COM port of the USB

- **How to Connect grblControl on an Arduino**

1) In the menu go into Service - Setting

2) And choose the serial port number that the Arduino / GRBL is connected to. The baud rate should be 115200 if you are using the latest version of GRBL

3) If it doesn't work make sure that you don't have another software connected to this serial port! (Disconnect it and try again)

- **States**

1) **State panel**

2) **Work coordinates**

   Represents current X, Y & Z local coordinates of the CNC.
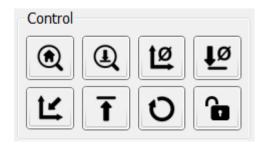
3) **Machine coordinates**

   Represents current X, Y & Z absolute machine coordinates.

4) **Status**

a) One of following CNC status:

b) Idle - waiting for a G-code command

c) Running - running a G-code command

d) Home - homing cycle is executing

e) Check - G-code command check mode is turned on

f) Hold - paused by a "!" command, need to be restarted by a "~" command

g) Alarm - CNC doesn't know where it is and blocks all G-code commands

h) Door - door sensor has triggered

- **Control**

  Control panel

1) ⌖ **Home button**

   Starts the homing cycle procedure with "$H" command

2) 🔍 **Z-probe**

   Starts the zero Z-axis search procedure using the
   command specified in the settings ("Z-probe commands"
   box).

3) Example command:

   G91G21; G38.2Z-30F100; G0Z1; G38.2Z-1F10

4) 🔲 **Zero XY**

   Zeroes the "X" and "Y" coordinates in the local coordinate
   system. Also retains an local system offset ("G92") for
   later use.

5) 🔲 **Zero Z**

   Zeroes the "Z" coordinate in the local coordinate system.

6) 🔲 **Restore XYZ**

   Restores local system coordinates with "G92" command.

7) ⬆ **Safe Z**

   Moves tool by "Z"-axis to safe position. Position
   coordinate can be specified in the "Safe Z" setting.

   Position must be specified in *machine* coordinates.
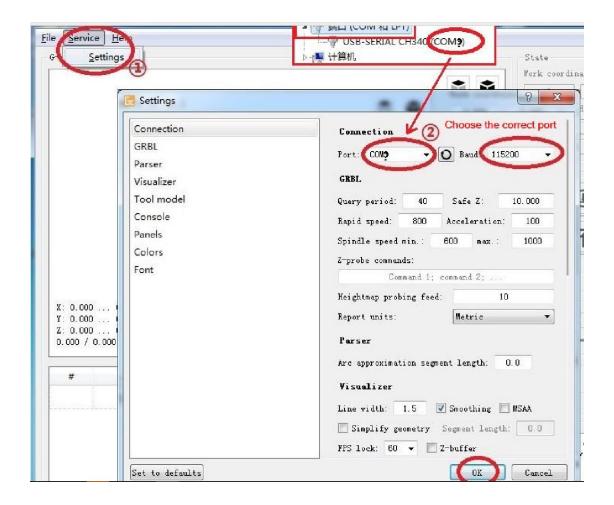
8) ↻ **Reset**

Resets CNC with "CTRL+X" command

9) 🔓 **Unlock**

Unlocks CNC with "$X" command.

- **Open grblControl software to connect the machine, choose the correct port：**

Connection succeeded：[CTRL+X] < Grbl v1.1 ['$' for help]

https://github.com/gnea/grbl/wiki/Grbl-v1.1-Configuration

Laser Engraving Instructions

● **Attentions：**

WARNING LASER FOCUS REDUCE HEAT AND GLARE WHICH CAN HARM HUMAN, ANIMALS AND OBJECTS. DO FOLLOW THE INSTRUCTION. MISUSE WILL BE AT YOUR OWN RISK.

- Avoid direct eye contact, may lead to blindness
- Avoid exposure to body surface, it burns
- Put base plate under the workpiece
- Avoid combustible object or gas.
- Keep it away from incompetent people, such as children or pregnant women
- Do NOT take apart the laser without instructions
- Do NOT use it on material that reflects the light
- Wear goggles while using
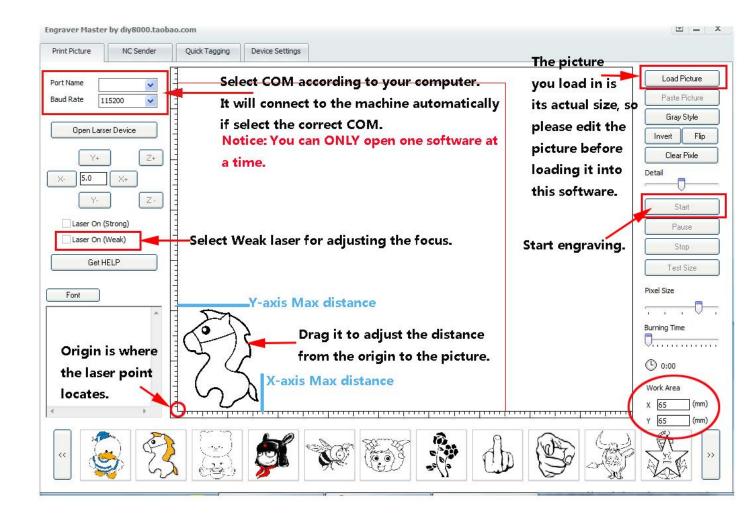- Turn off the power when not use

Non-working materials：

It cannot work on any metal, ceramic, jewelry, etc. Also, it cannot work on any material that reflects light or is transparent.

● **Software：Miniature Steward**

1. Main interface

- **Engraving Position:**

2. Turn on the weak light, it will appear a point on the work piece. Then swivel the knob to adjust the point to the smallest.

Rotary knob for adjusting the focus