

# CNC1310

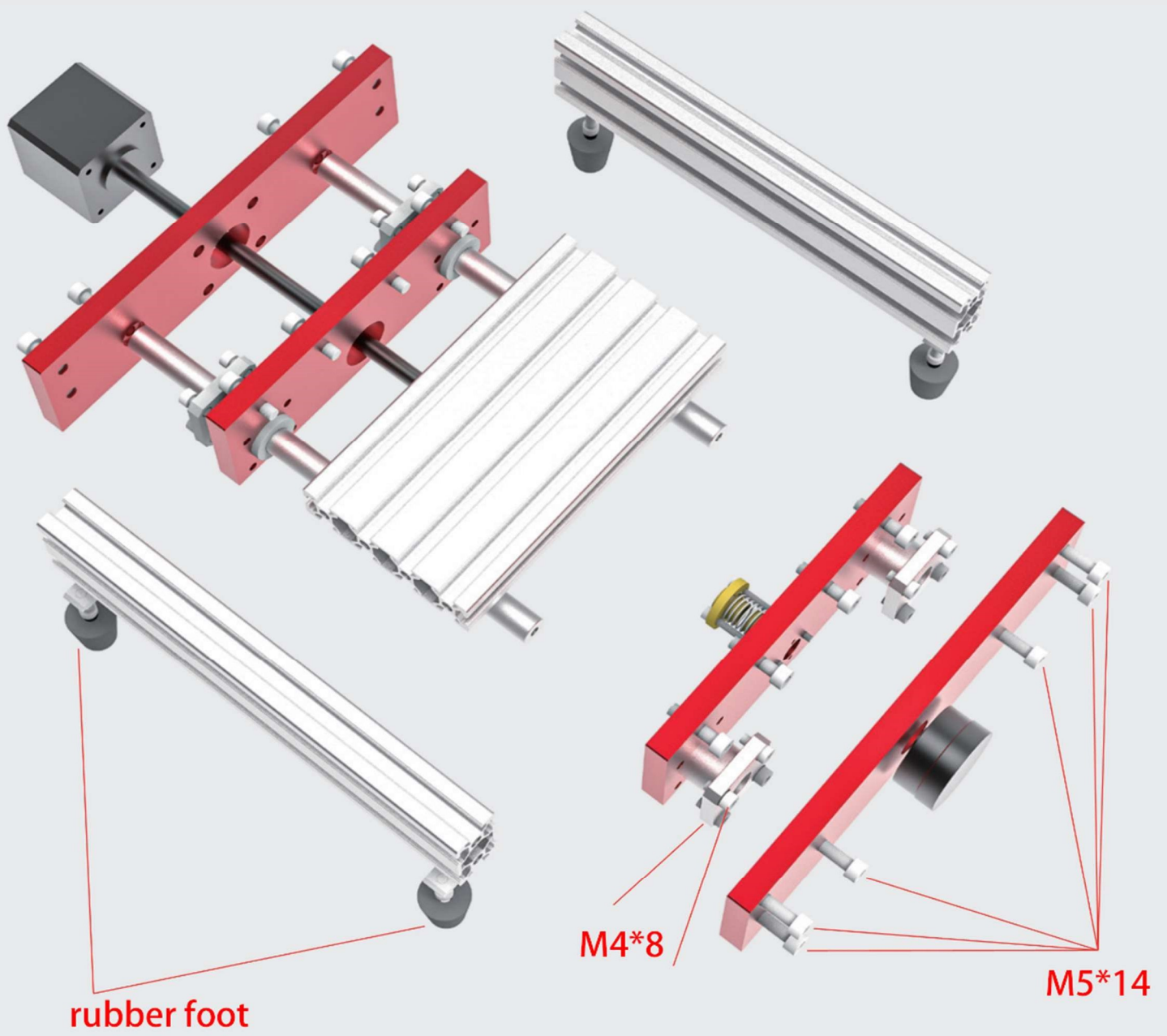
## Installation and instruction



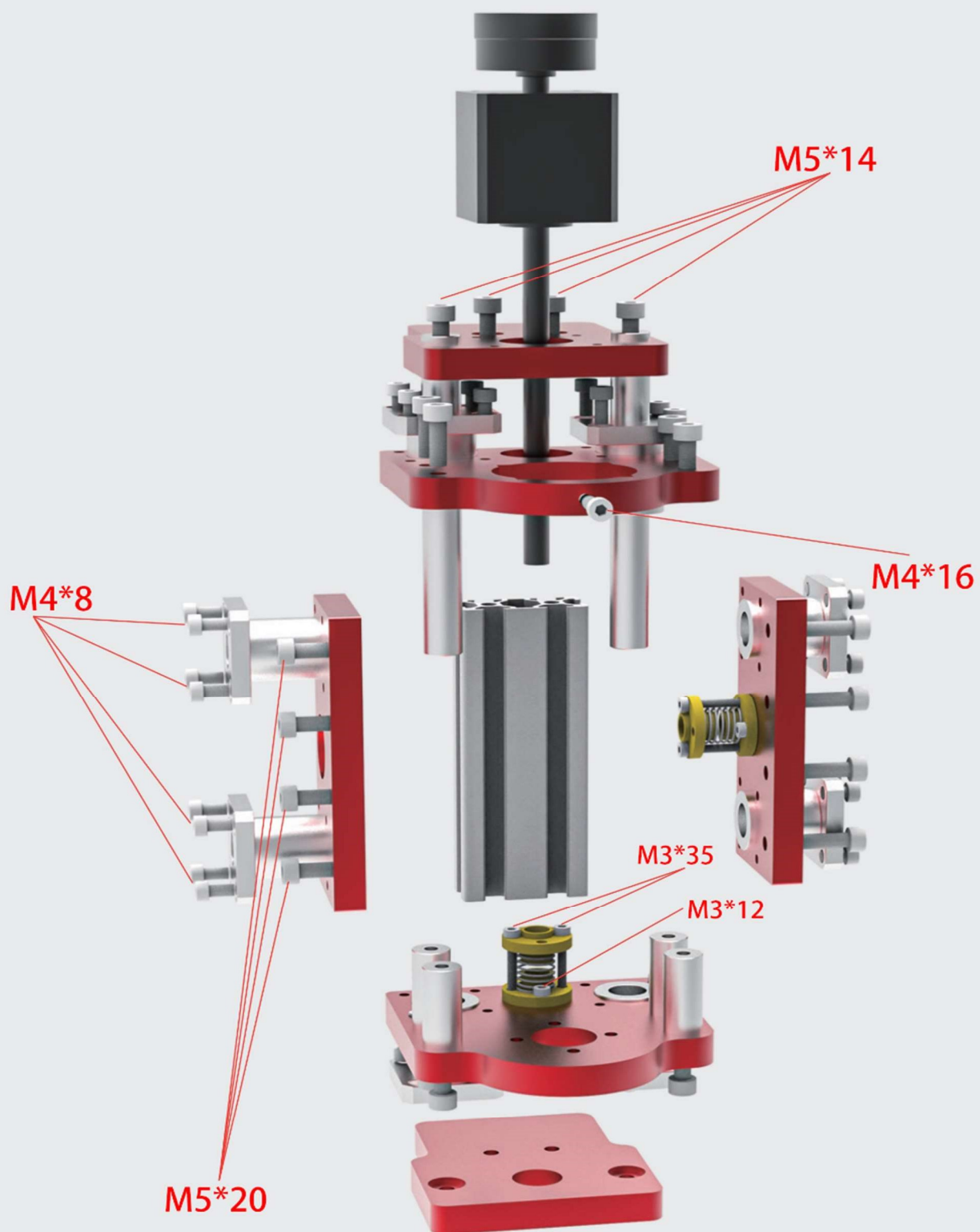
Accessories list:

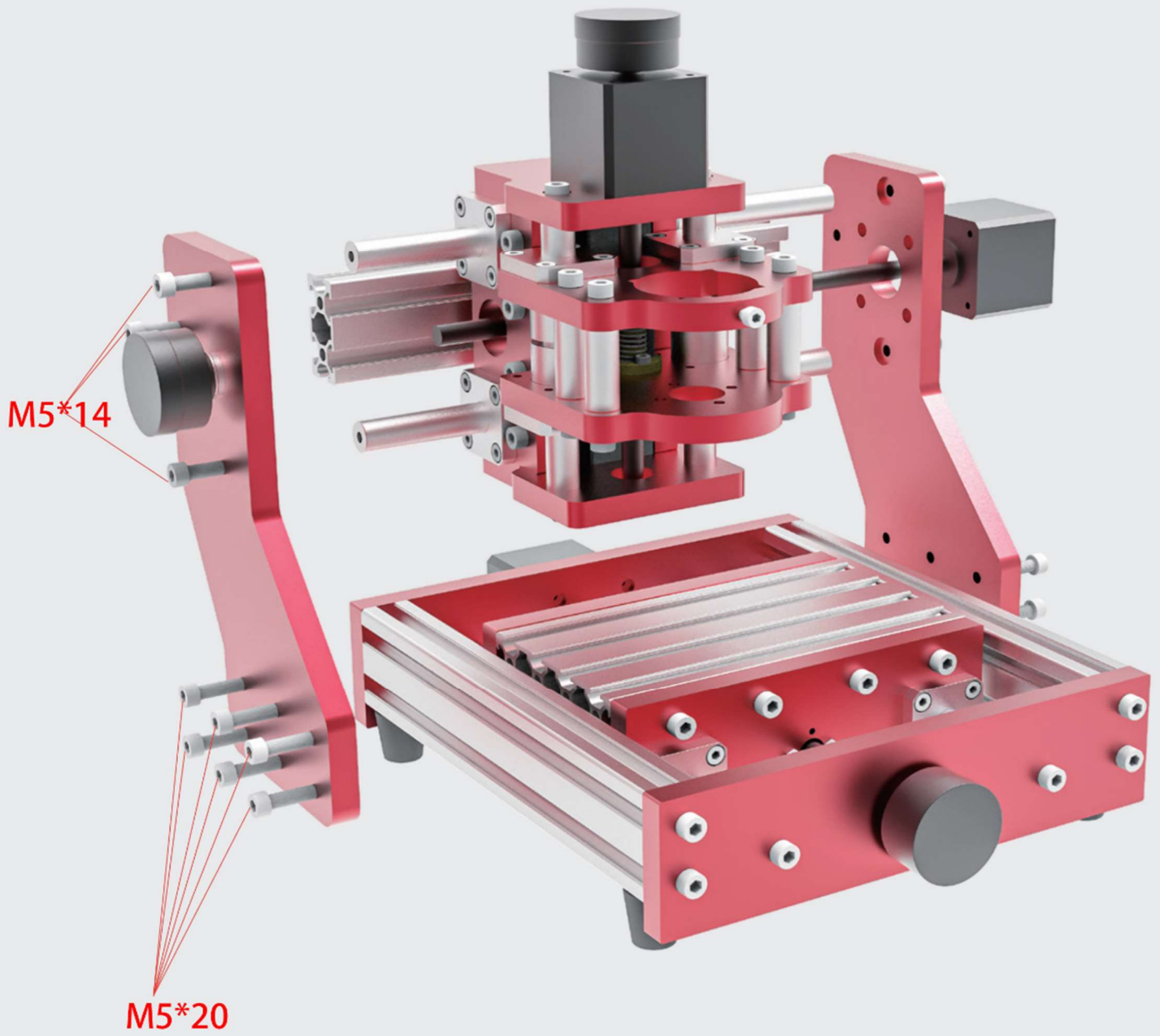


STEP1. Base Assemble

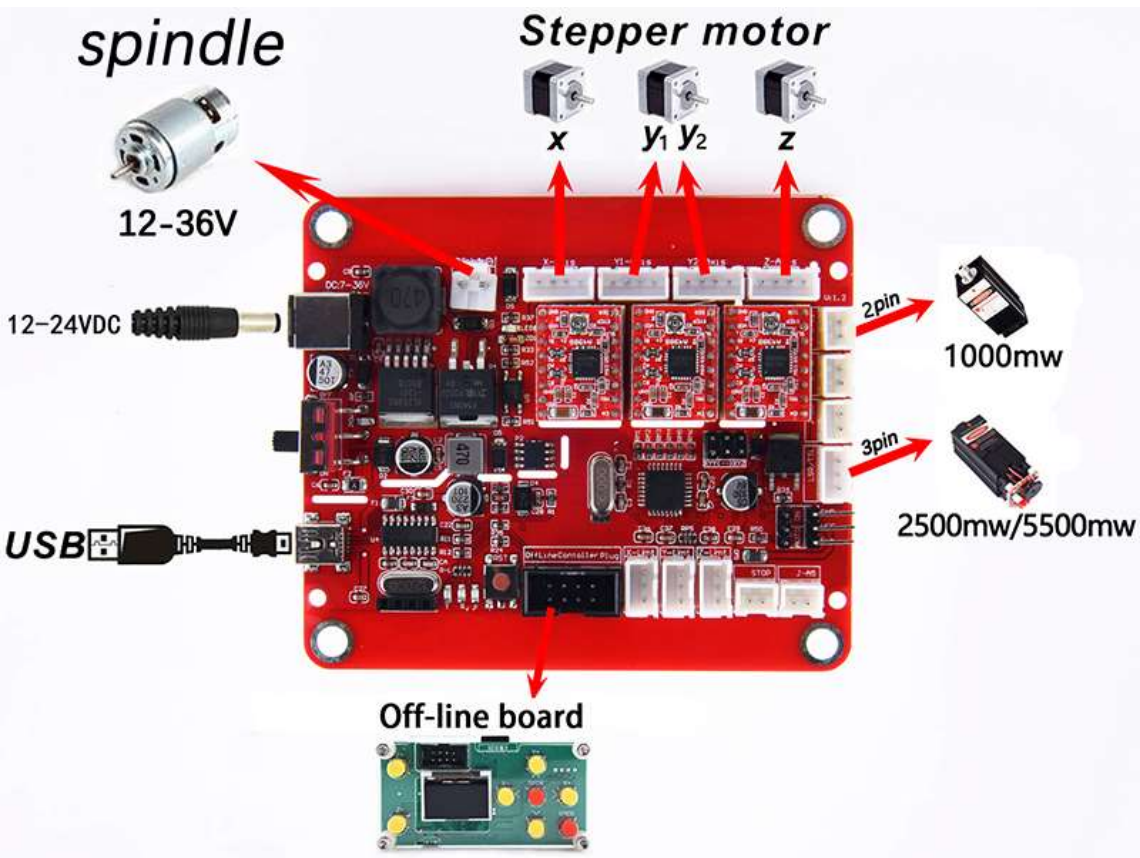
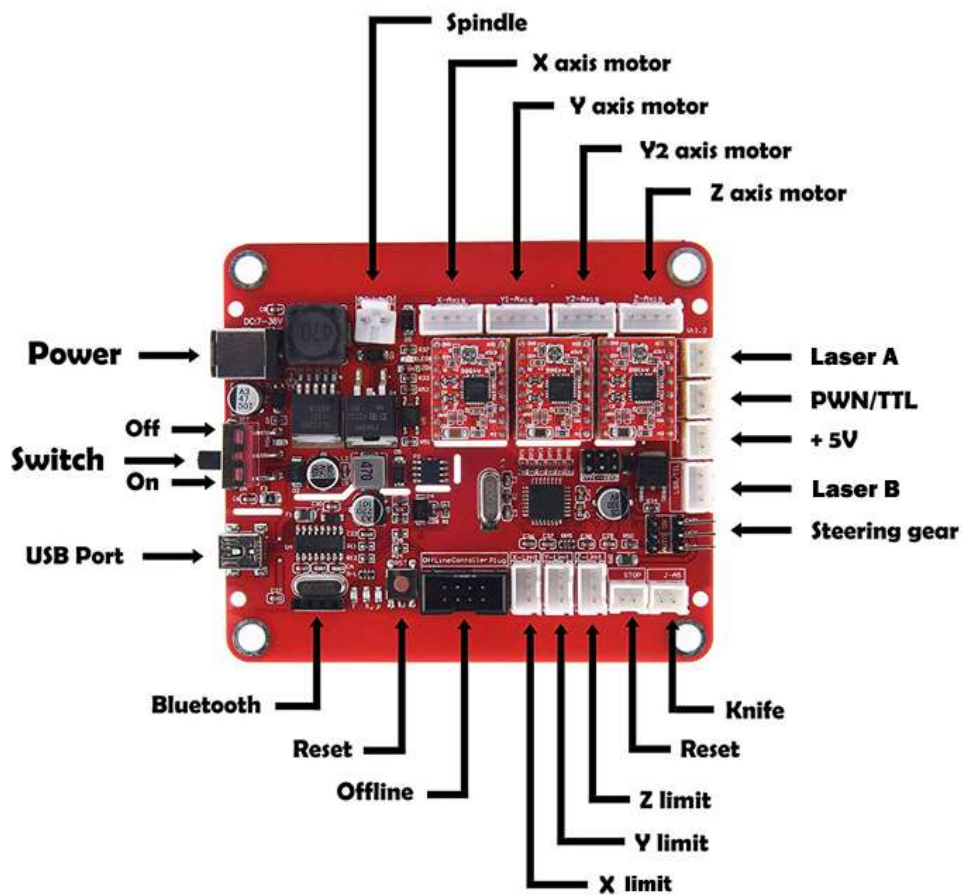


## STEP2. Z Axis Assemble





# Control board

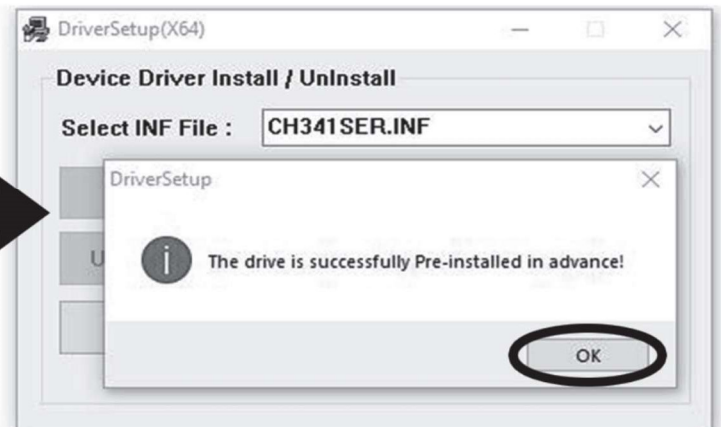
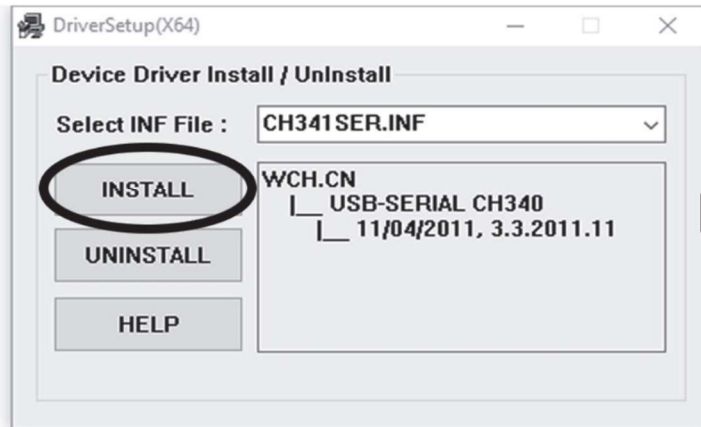


# Driver install:

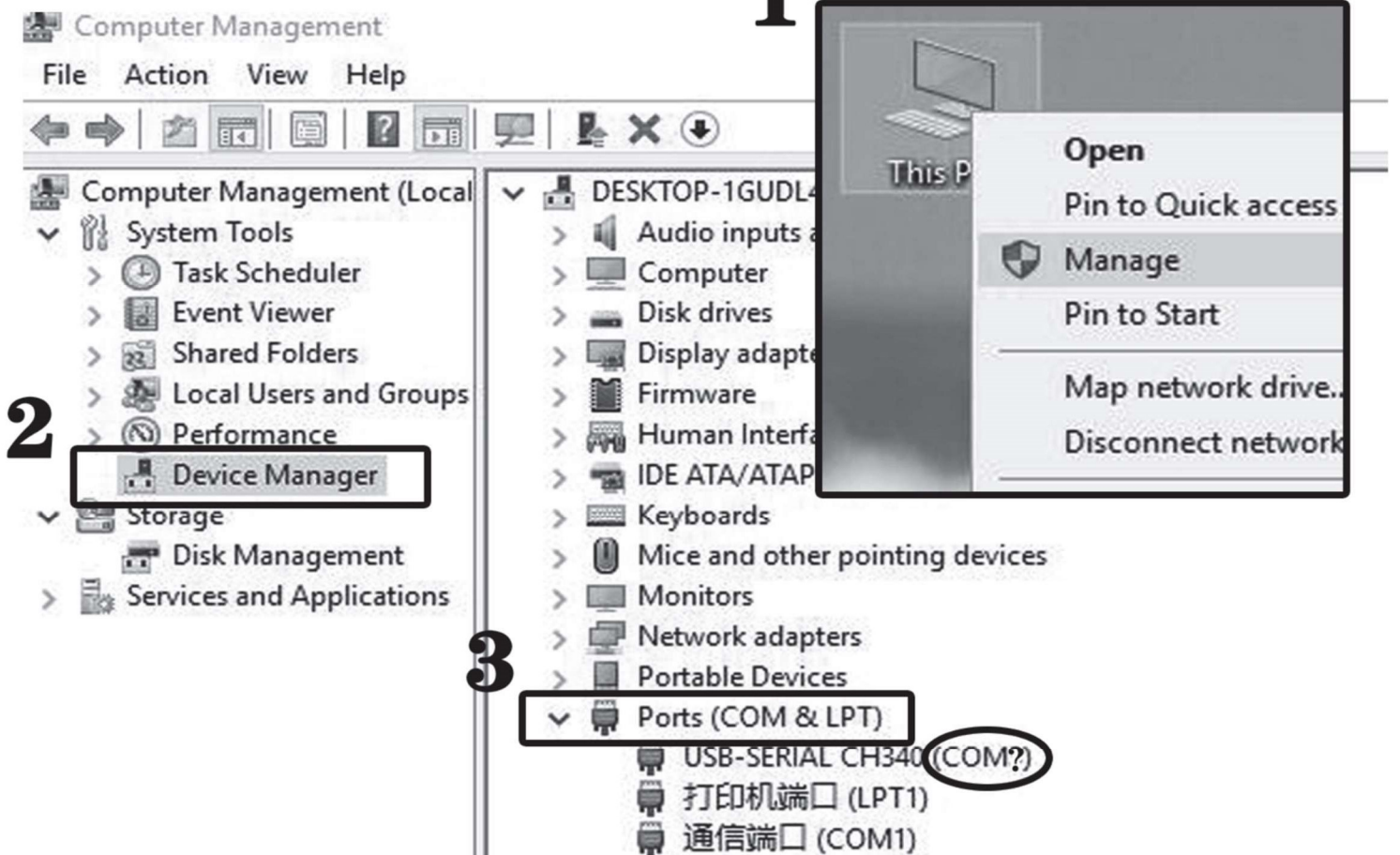
- Install CH340 drive Program.

Double-click CH340SER --Click Installation— Wait for the installation program to end.

(This drive program Supports XP32/64WIN7 32/64)



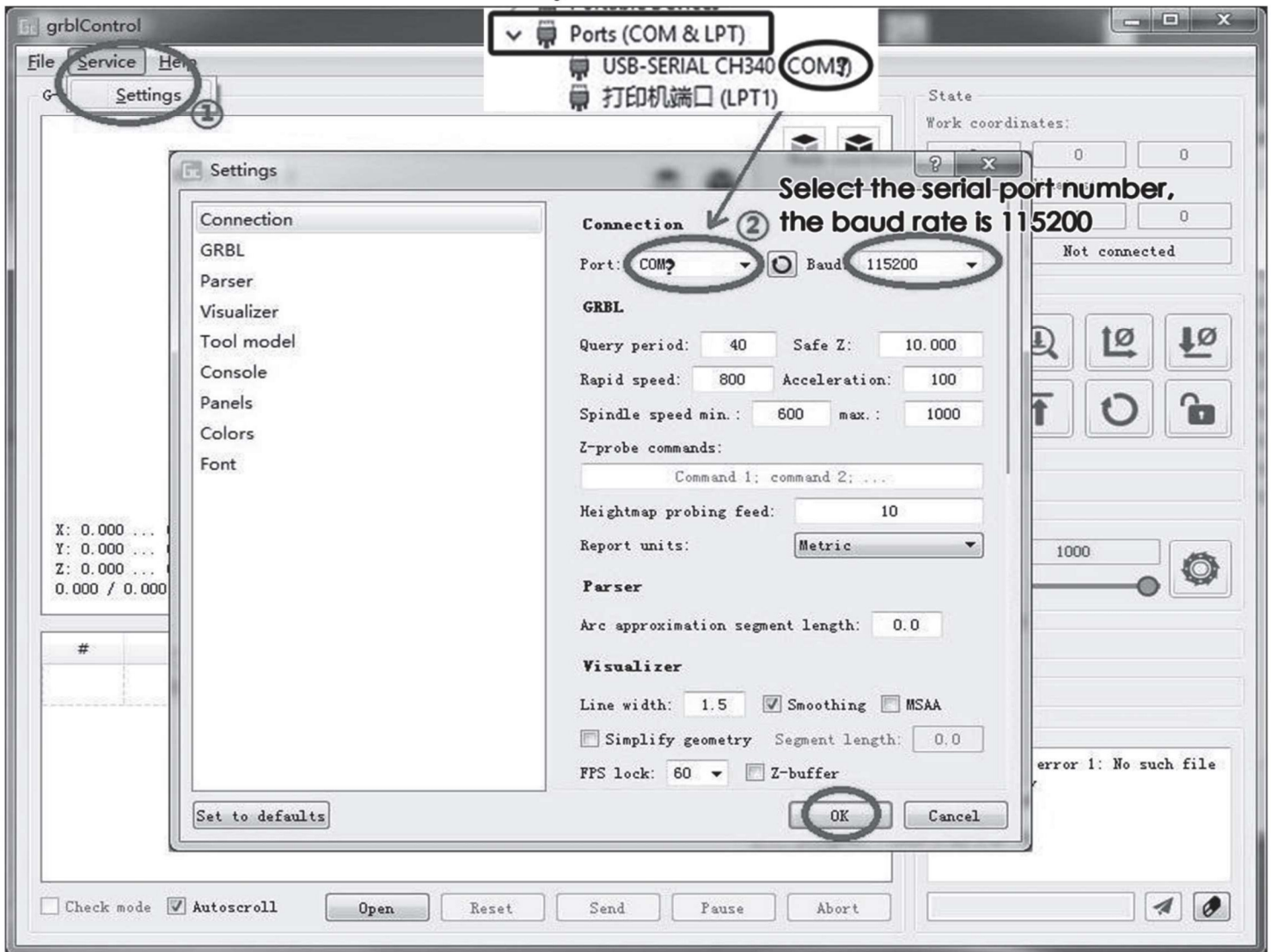
- Right-click My Computer-Manage-Device Manager-Ports(COM&LPT), check whether there is CH340 Port number. If installation success will appear COMx, remember the number after COM.



- Use Grblcontrol(Control Software) to connect machine:



Open Control Software Grblcontrol. This software is for control CNC 3 Axis carving. First of all set port number, and then return to the main interface of software, software will connection machine automatically.

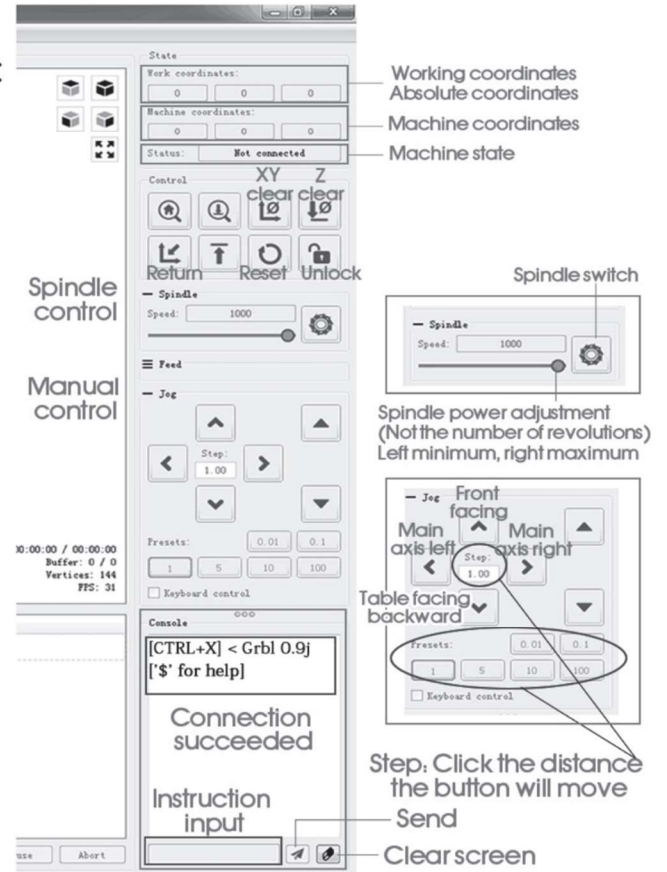
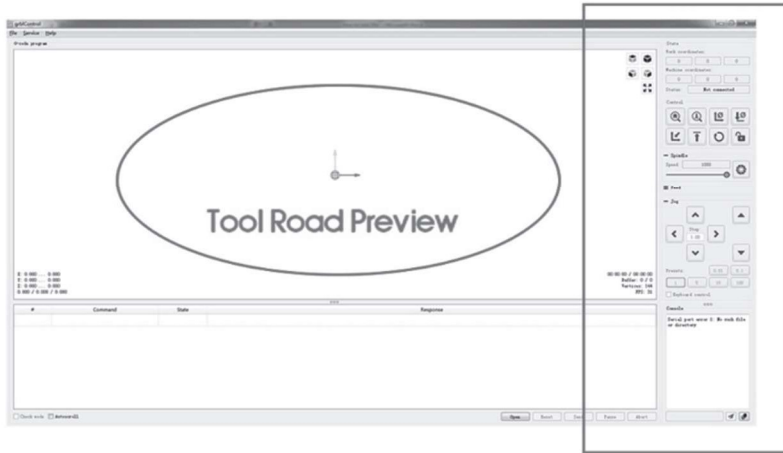


The port number must be the same as last step, otherwise software can not connect the machine.

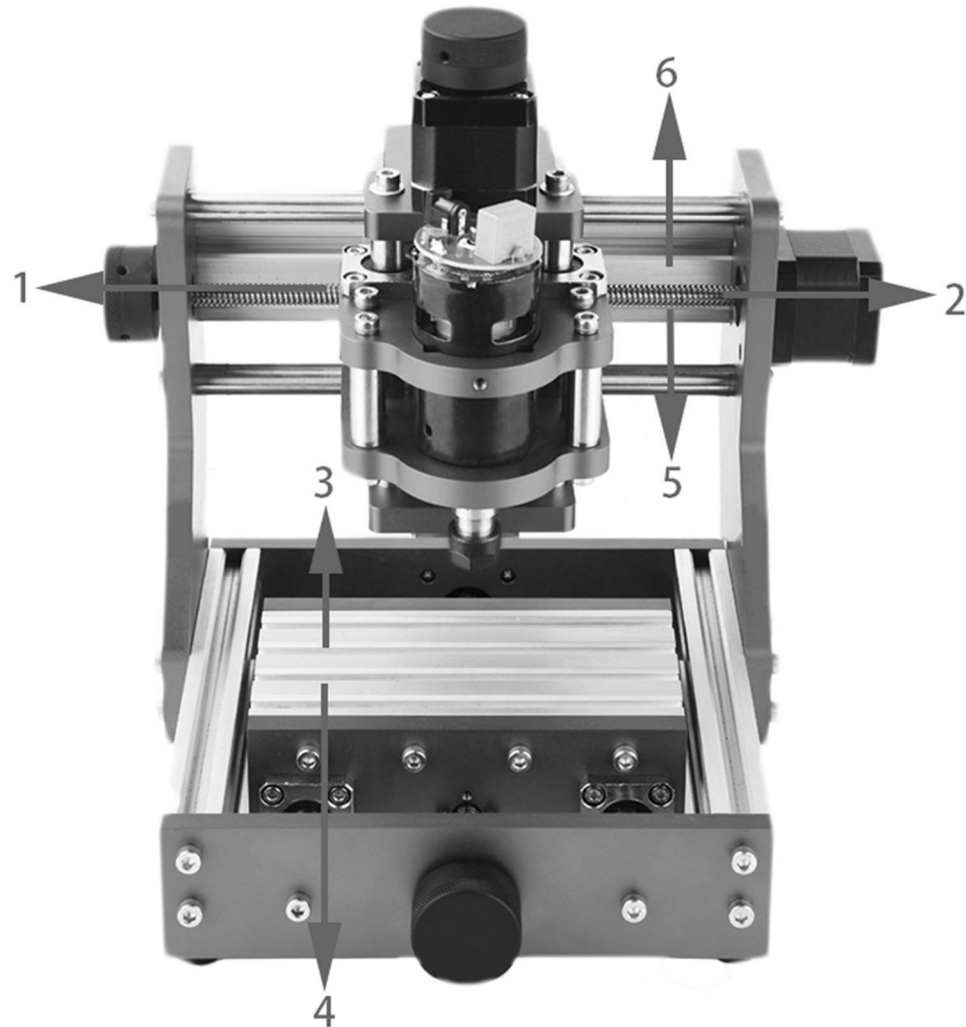
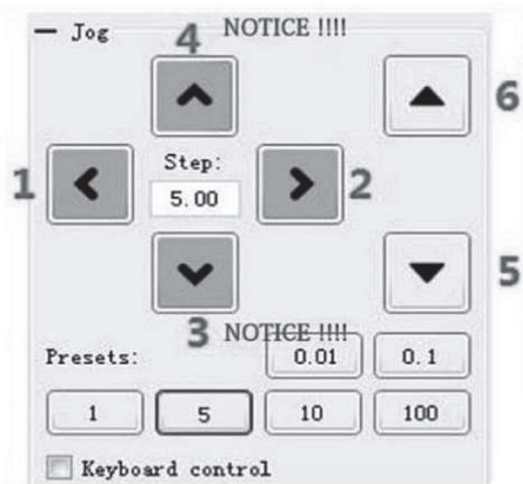
If connect machine successfully the main interface will display: [CTRL+X] < Grbl 0.9j ['\$' for help]. If no this display or display other information it means connection failure. Please check whether you have followed all the steps. Or close software, pull out USB wire. And connect USB wire-open software-set port number-connect

machine again.

● Grblcontrol The main interface introduction:



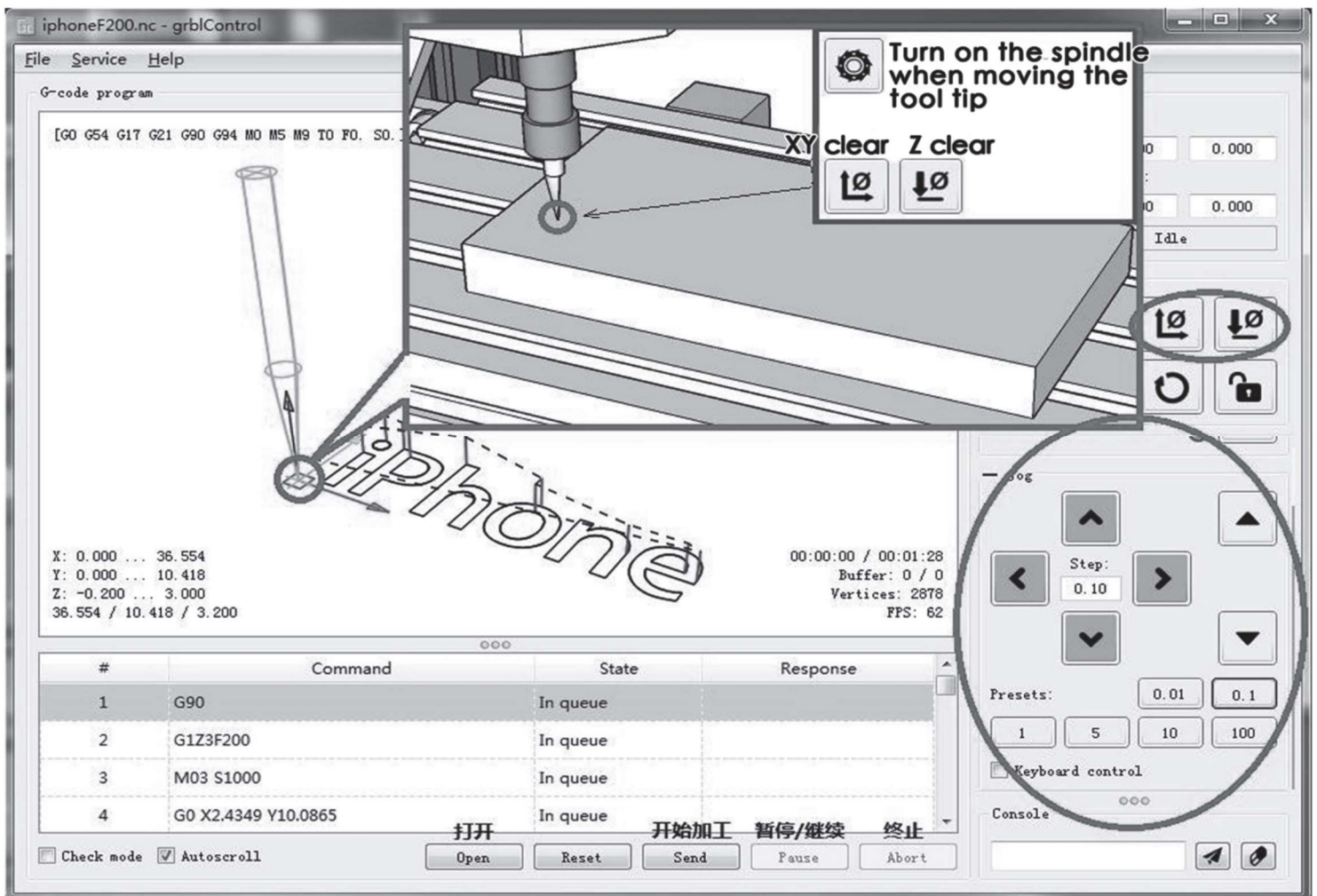
● Direction check: check whether the machine moves the same direction when you click on the software



If you click the button, The machine is not moving in the corresponding numerical direction, that means the stepper motor in this direction is reversed . Please exchange the motor wire on the control board.

3 .After direction checking we can start to work.

- Cutter setting: to set the position where we start to work. Move cutter to the position, click xy and z axis coordinate zero clearing, then it will start to engrave at this point.
- When you do last step please make the spindle running because the cutter may touch the object you want to engrave.



After first testing, then you can try to make your own gcode for engraving.

Gcode is the command that make the machine move. It should be a .nc file. In the folder there are some .nc files. You can check.

Matched software to make .nc file is ARTCAM(In the link it is old version, if you

want to get latest version you can search it on internet, it is very common software):

<https://www.dropbox.com/s/n1pi86z2lu5zd8q/ArtCAM.rar?dl=0>

This is one version, you can also get some other latest version on internet.

You can also watch these videos on Youtube to know how to make .nc files by using ARTCAM:

<https://www.youtube.com/user/delcamartcam/videos>

## How to set the Moving Direction of the Axis:

Test the machine after assemble.If the axis runs in a wrong dirction,please after the settings as the picture below shows.

Engraver Master

Print Picture NC Sender Quick Tagging Device Settings

Port Name COM1  
Baud Rate 115200

Close

Y+ Z+  
X- 5.0 X+  
Y- Z-

Laser On (Strong)  
 Laser On (Weak)

Get HELP

Font

Load Pi  
Paste Pi  
Gray S  
Invert  
Clear P

Detail  
Star  
Paus  
Stop  
Test S

Pixel Size  
Burning Time

2. Set

1. Make sure the motherboard is connected to the computer successfully.

Engraver Master

Print Picture NC Sender Quick Tagging Device Settings

My Machine ?

Apply machine default settings

Default Acceleration:  
Default Moving Speed:  
XY Steps:

Friendly Machine List

八千电子  
DIY8000 COREXY1818 Desktop Laser

八千电子  
DIY8000 1720/4050 Desktop Laser

Laser Optimization  
Acceleration, mm/sec<sup>2</sup>  
20  
Moving speed (mm/min)  
500  
X STEPS Y STEPS Z STEPS  
200 200 200  
 Reverse X axis  
 Reverse Y axis  
 Reverse Z axis  
 Use as a Pen Drawing Machine

Save Settings

Gribl Settings >>>

1

2

3