

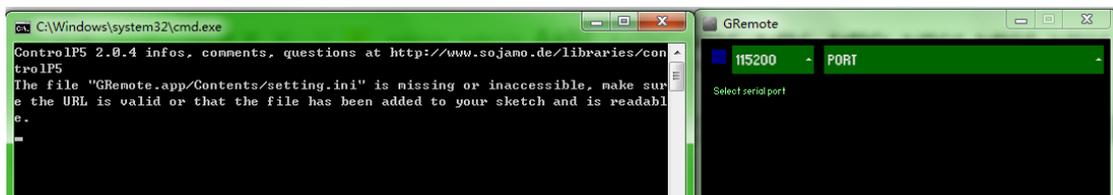
## XY Plotter V2.0 software manuals

### A. Preparations

1. An assembled XY Plotter V2.0
2. A personal computer
3. Arduino IDE
4. GRemoteFull: the package of G-code examples, firmware of Me baseboard and program for controlling.

### B. Installation and setup of the softwares

1. Download package GRemoteFull from <https://github.com/Makeblock-official/XY-Plotter-2.0> and extract it.
2. Open file "GCodeParser.ino" in "GRemoteFull/GCodeParser" with Arduino IDE.
3. Connect Me baseboard to your PC and set the COM port 11. (The serial port may be different on different computers)
4. Click Tools->Board on the menu of Arduino IDE and choose "Leonardo".
5. Click the "Upload" icon, and Arduino IDE will compile file "GCodeParser.ino" and upload it to Me baseboard automatically.
6. Close Arduino IDE.
7. Double-click file "GRemote.bat" in "GRemoteFull" and it will display the following windows.



8. Set the COM port, then GRemote will show you the UI as follows. That all for this part. (See Fig. 1)

Fig. 1

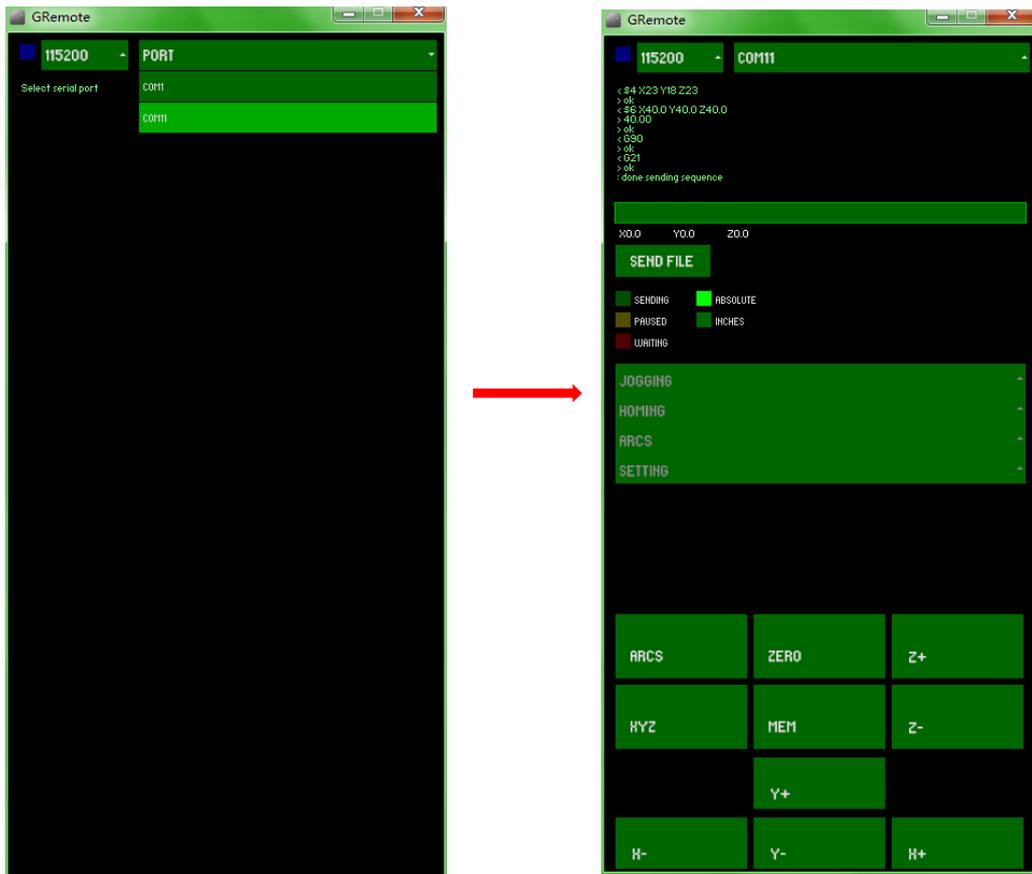


Fig. 1

### C. User guide of GRemote UI

From now on, you can control XY Plotter V2.0 with your mouse and keyboard. The motion of X-axis and Y-axis can be controlled by direction keys. The motion of Z-axis, namely the 9g-servo, can be controlled by key '<' and '>'. Some frequently-used functions are mapped to hotkeys, such as 'J', 'H', 'A', ';', '[', ']', 'x', 'y', 'z', 'shift+x', 'shift+y', 'shift+z', etc. The default pin setting matches the standard combination. If you assemble the plotter in your own way, please confirm the pin setting. The menu of pin setting is under the "SETTING" item. The following pictures show you some tips for GRemote UI. (See Fig. 2)



Fig. 2

a) Main window (See Fig. 3)



Fig. 3

b) Introduction of functional areas (See Fig. 4- Fig 8)

1

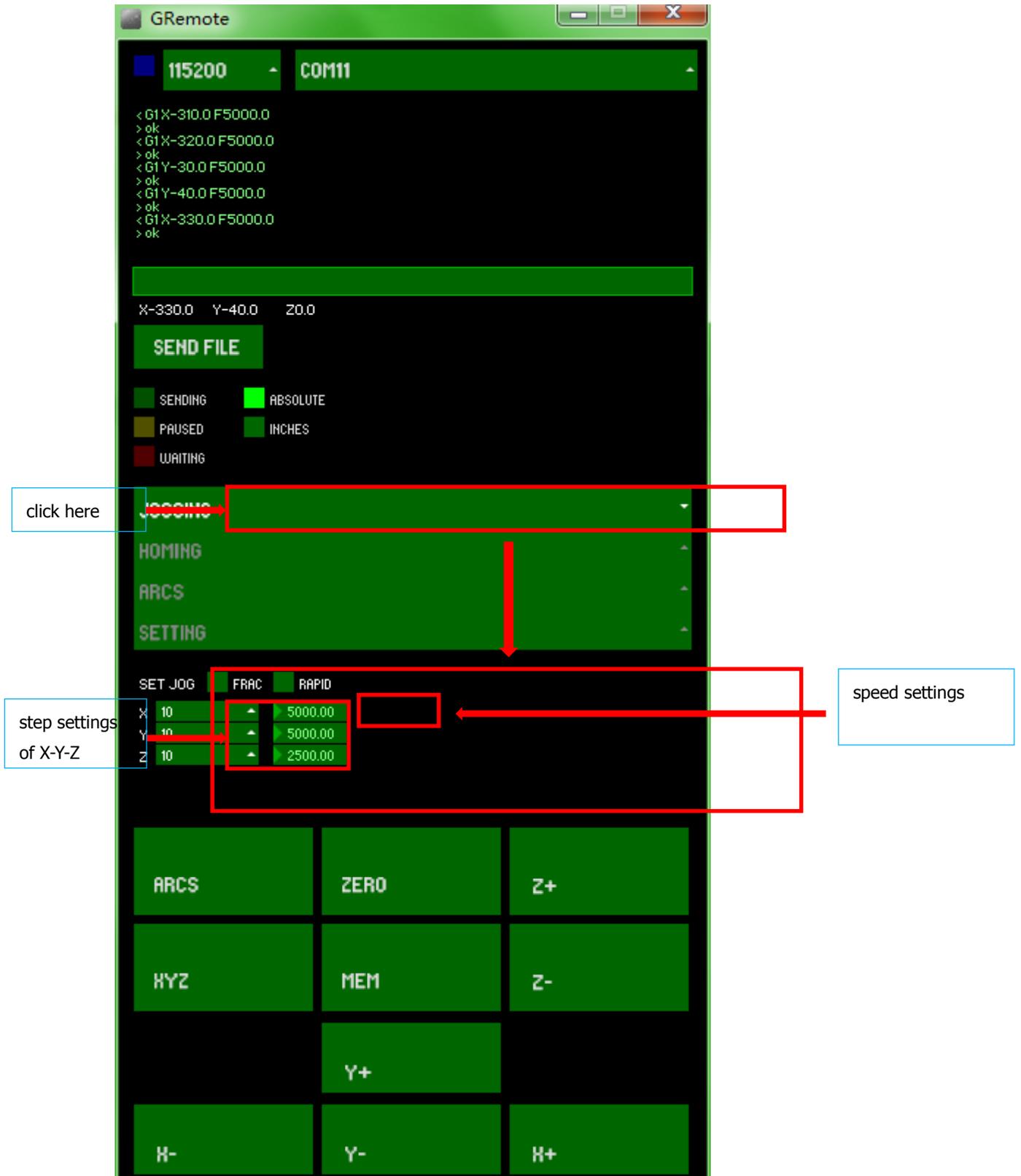


Fig. 4

2.



Fig. 5

3.



Fig. 6

4.



Fig. 7

5.



Fig. 8

## D. Do drawings with GRemote

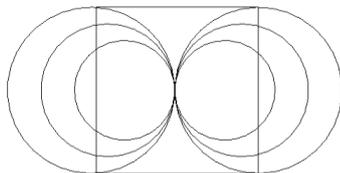
Turn on the power of Me baseboard, then reset the X-axis, Y-axis and Z-axis. Click button "SEND FILE" and choose a G-code file (\*.cnc). Then the plotter will run automatically.

Here are two simple examples of G-code files. You can copy the text into a text editor and save it as a \*.cnc file.

### Example 1:

```
%  
G21 (All units in mm)  
G00 Z20.0  
G00 X50.0 Y50.0 F3500.0  
G01 Z33.0 F2500.0 (Penetrate)  
G01 X100.0 Y50.0  
G01 X100.0 Y100.0  
G01 X50.0 Y100.0  
G01 X50.0 Y50.0  
G00 X75.0 Y75.0 Z20.0  
G01 Z33.0  
G02 X75.0 Y75.0 I25.0  
G02 X75.0 Y75.0 I20.0  
G02 X75.0 Y75.0 I15.0  
G03 X75.0 Y75.0 I-25.0  
G03 X75.0 Y75.0 I-20.0  
G03 X75.0 Y75.0 I-15.0  
G00 Z20.0  
G00 X50.0 Y50.0 F3500.0  
%
```

The picture of example 1:



### Example 2:

```
G21  
G90 Z1.000  
G00 X0.000 Y0.000  
G00 Z13.000  
F2500  
G00 X75.231 Y62.747 Z20.000
```

```

G01 X75.231 Y62.747 Z33.000 F2500
G01 X48.645 Y44.514 Z33.000 F2500
G01 X57.762 Y75.439 Z33.000
G01 X32.206 Y95.090 Z33.000
G01 X64.435 Y95.975 Z33.000
G01 X75.226 Y126.353 Z33.000
G01 X86.027 Y95.975 Z33.000
G01 X118.253 Y95.099 Z33.000
G01 X92.700 Y75.439 Z33.000
G01 X101.825 Y44.520 Z33.000
G01 X101.823 Y44.514 Z33.000
G01 X101.817 Y44.514 Z33.000
G01 X75.231 Y62.747 Z33.000
G00 X0.000 Y0.000 Z13.000
G00 Z13.000

```

The picture of example 2:



XY Plotter V2.0 can draw any standard G-code file, no matter how complicated it is.

#### **E. Software to generate G-code**

You can use commercial software, such as MastCAM and Artcam, to generate G-code. Open source software is also a good choice. Here are some open source softwares we recommend to you.

1. [InkScape+gcodetools.tar.gz](#) Open Source vector graphics creation software+gcodetools plug-in
2. [dxf2gcode](#): useful for engravings
3. [FreeCAD](#): 3D CAD design program; G-Code generation partly integrated (2011)
4. [HeeksCNC](#): add-on for [HeeksCAD](#); requires STEP files instead of STL; activity partly abandoned by its main author (2011)
5. [cam-occ](#): based on solid models instead of trimeshs (no recent releases, but development seems to be active (2010))
6. [monocam](#): no official releases, yet; no recent development activities (2010)
7. [camvox](#): no official releases, yet; no recent development activities (2010)
8. [gcncam](#): uses DXF as input; quite active

9. [CNC Code Generator](#): java-based, recently not very active (2010)

10. [GCAM](#): DXF or GERBER input; sparse documentation

#### External software lists

1. [EMC-Wiki](#): exhausting list of CAD/CAM/conversion software

2. [xtronic wiki](#): broad overview of CAD/CAM software

3. [Simple G-Code generators](#): a list of small scripts and simple interface for generating basic toolpaths for EMC2

You can also download the softwares you need in the following websites.

1. [http://wiki.linuxcnc.org/cgi-bin/wiki.pl?Cam\(Listof CAM references](http://wiki.linuxcnc.org/cgi-bin/wiki.pl?Cam(Listof%20CAM%20references))

2. <http://cadcamcae.wikia.com>

## F. Generate G-code with Inkscape and gcodetools

1. Download Inkscape and gcodetools.tar.gz.
2. Install Inkscape.
3. Extract the gcodetools package to path "D:\Program Files (x86)\Inkscape\share\extensions", which may differ according to the path to install Inkscape. Launch Inkscape and you'll see gcodetools in the plug-ins.
4. Create a new file and a layer.
5. Generate G-code from a vector graphic or a bitmap.
  - 5.1. Vector graphic: Open a \*.svg, \*.dxf, or other vector graphic file with Inkscape, or create a new vector graphic in Inkscape.
  - 5.2. Bitmap: Import a bitmap (\*.jpg, \*.png, \*.tiff, \*.pdf, etc.). Click "Path->Trace Bitmap" on the menu, and set the parameters to generate a vector graphic from the bitmap. Delete the origin bitmap.
6. Select the vector graphic and Click "Object->Ungroup" on the menu.
7. Click "Path->Stroke to Path". Click "Path->Object to Path".
8. Click "Extensions->Gcodetools->Orientation points...", check "3-points mode" and click the "apply" button.
9. Click "Extensions->Gcodetools->tool library...", check "Lathe cutter" and click the "apply" button.
10. Click "Extensions->Gcodetools->Path to gcode...". Set the tolerance, maximum depth, Z-axis parameters, filename and path of G-code file, then click the "apply" button.
11. Modify the suffix of the generated G-code file to ".cnc". Open the G-code file with a text editor and do the following editing jobs. Set the starting position of Z-axis to Z15.0, working position to Z35.0, F code value to F100.0-F20000.0. Confirm the parameters above and Open GRemote to draw the picture.

## G. Click here for more details.

<http://www.makeblock.cc/xy-plotter-robot-kit-v2-0/>