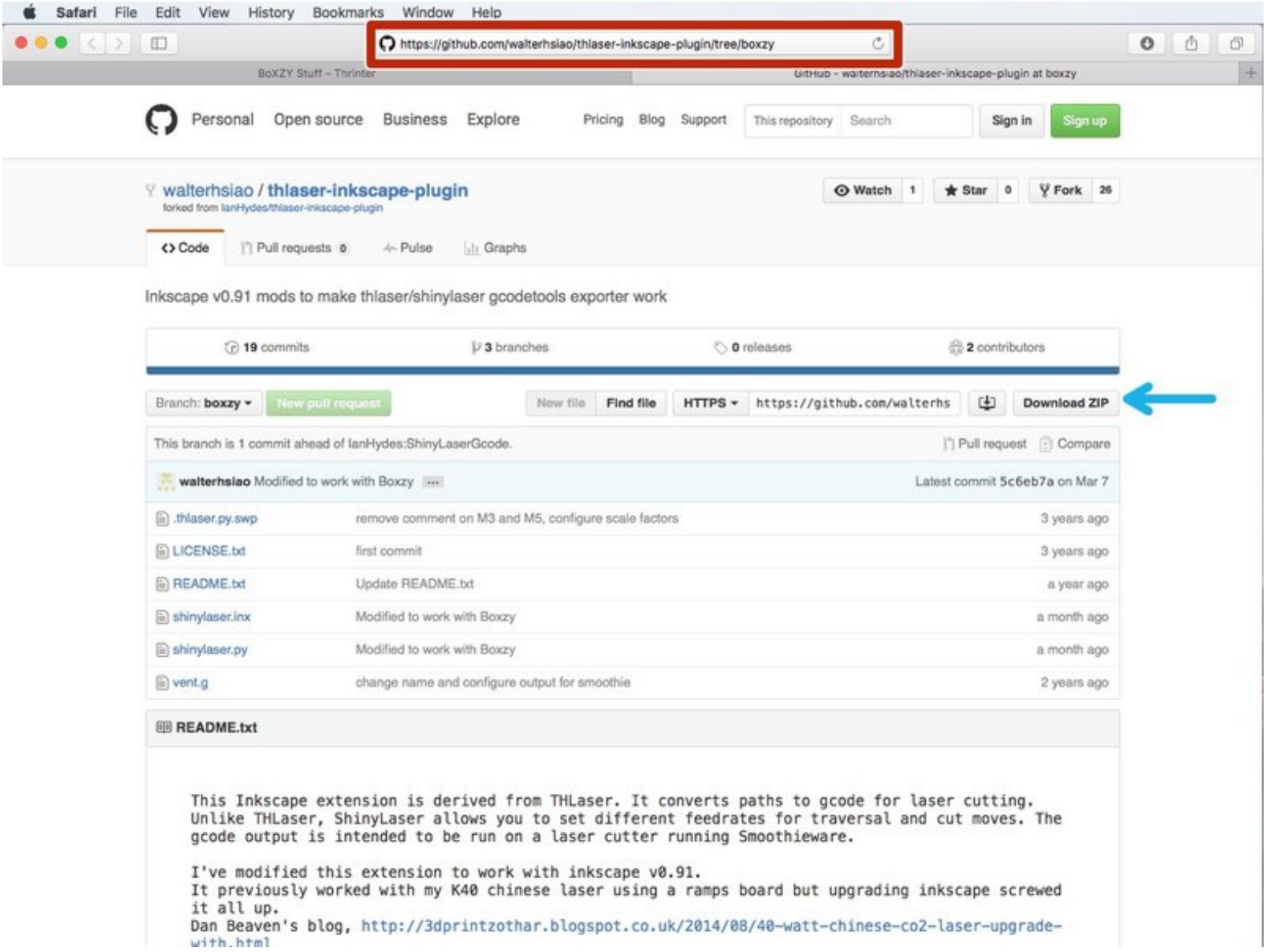
Laser etching control solutions for Mac OS X and BoXZY

This guide will show you how to install a plugin into Inkscape and how to create Gcode files for Mac OS X users to use their BoXZY laser.

Written By: BoXZY

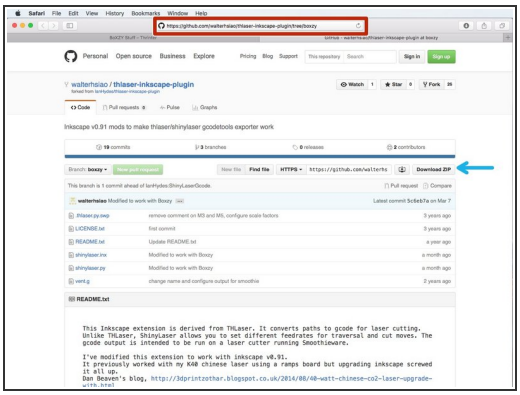


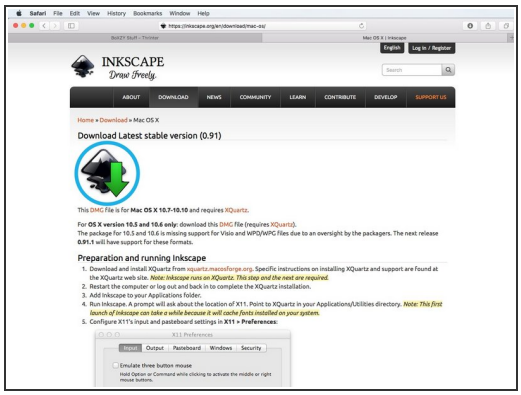
This document was generated on 2019-10-02 09:52:04 AM (MST).

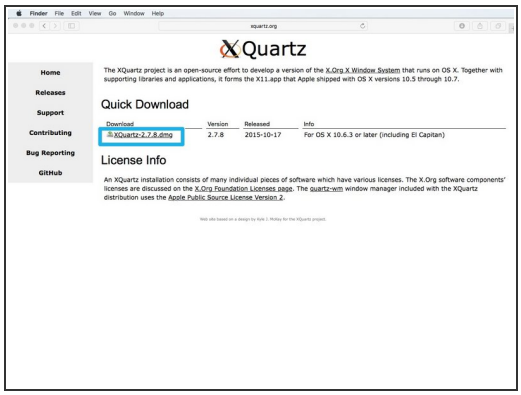
INTRODUCTION

Thanks to Thrinter.com and walterhsiao on Github.com we have a method for Mac OS X users to operate the BoXZY laser with Gcode produced in Inkscape, an open source vector software. Windows users can also install Inkscape and the same plugin to produce Gcode.

Step 1 — Downloading the Inkscape software and extension plugin

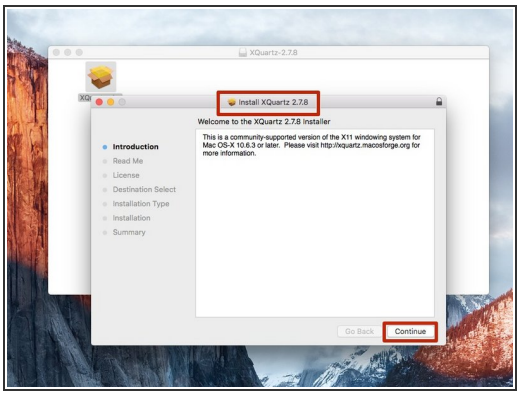


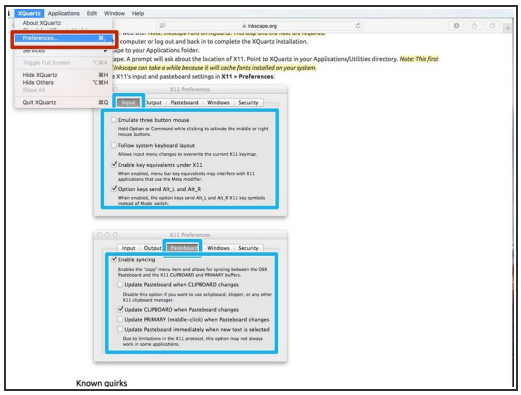


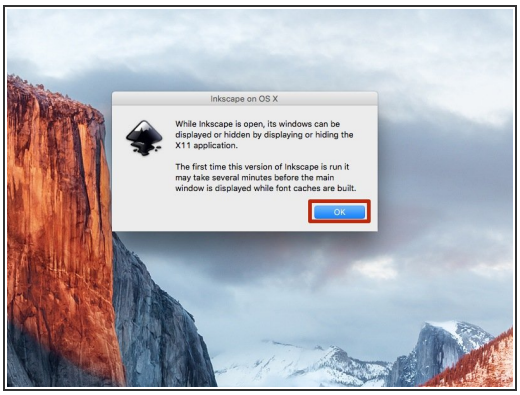


- In your web browser go to this link and download the **thlaser-inkscape-plugin ZIP** file from github containing the Inkscape plugin, indicated by the blue arrow in the image
- In your web browser go to this link and download **Inkscape** for Mac, as indicated by the blue circle in the image
- In your browser go to this link and download **XQuartz**, as indicated by the blue box. XQuartz is required for OS X users to use Inkscape

Step 2 — Installing the software

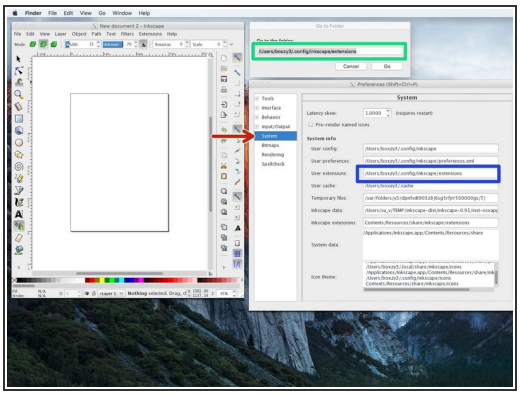


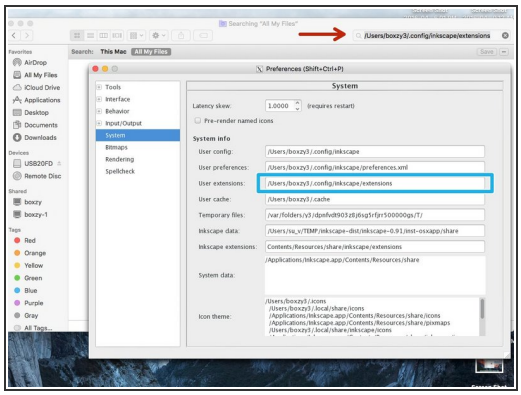


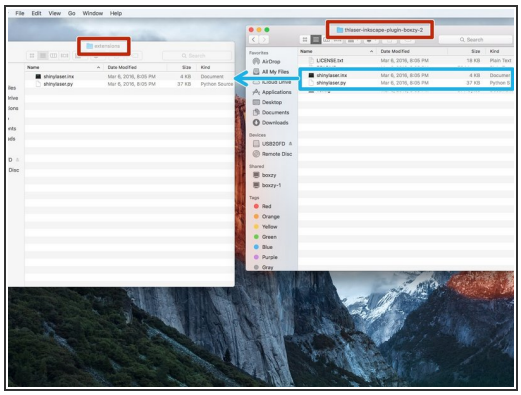


- Open **XQuartz** from where your browser downloads and follow the *Install XQuartz* installer procedures
- Run the newly installed **XQuartz** application. Select *preferences* from the XQuartz main menu and match the **X11 Preferences** tab contents as indicated by the blue boxes in the image, this information can also be found at this link, the official Inkscape website
- The final step to installing *XQuartz* is **restarting your computer**, do so now and return to this guide and finish this step
- When your computer is finished restarting open and run **Inkscape** from where your browser downloads. Note that it may take some time for the Inkscape window to open, please be patient

Step 3 — Installing the plugin into Inkscape

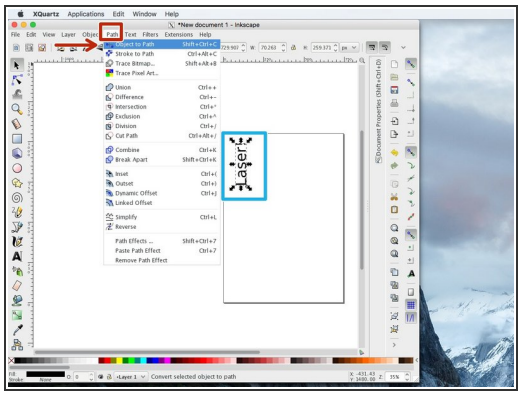


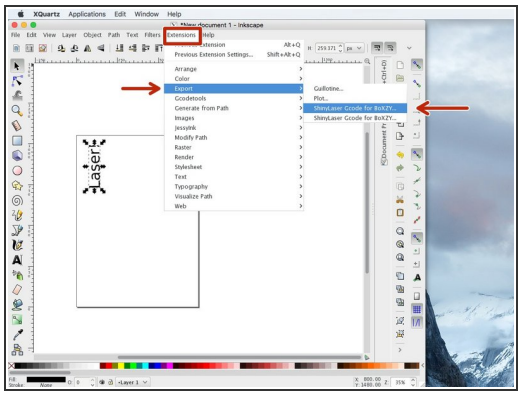


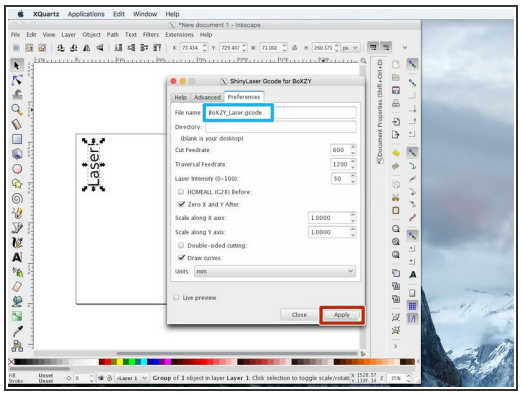


- Once the **Inkscape** window opens press *Shift+Ctrl+P* shortcut to open the **Preferences** window open then select **System** from the menu on the left hand side of the Preferences menu
- Copy the **User Extensions** address as indicated in the image with a blue square. Press *Cmd+Shift+G* to open the Go To Folder search window and paste the **User Extensions** address, as indicated with the green square. Select *Go* and the User Extensions folder for Inkscape plugins will open
- Open your web browser downloads and open the **thlaser-inkscape-plugin-boxzy-2** folder. Copy **shinylaser.inx** and **shinylaser.py** and paste them into the Inkscape User Extension folder you opened in the previous action in this step, this is indicated by the blue box and arrow
- Close **Inkscape** and then reopen it.

Step 4 — Inkscape export to Gcode

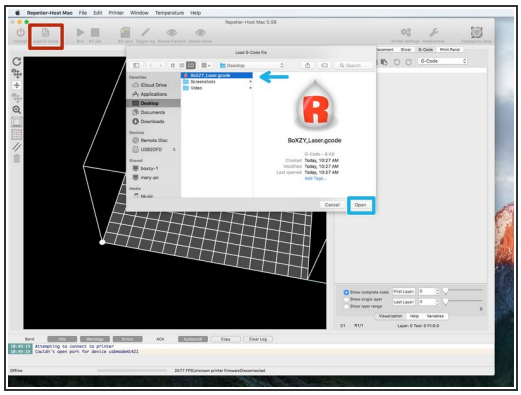


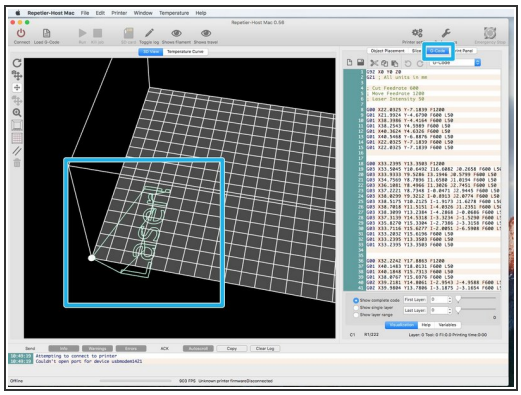


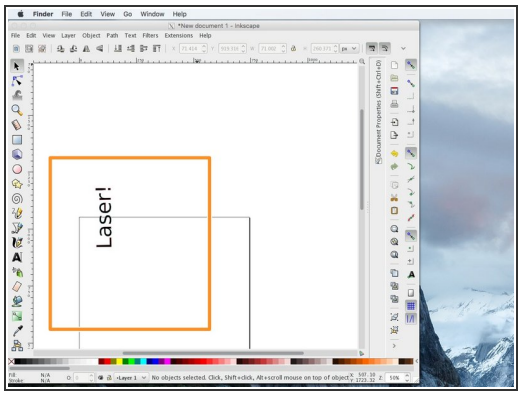


- With Inkscape open, draw your vectors making sure that stroke of the vector is in pixels and the color black. Select the vector(s) as indicated by the blue box in the image.
- With the vector(s) selected, go to the **Path Tab**, indicated by the red square and select **Object to Path**, indicated by the red arrow
- With the vector(s) selected, go to the **Extensions** tab, indicated by the red square and select **Export** from the menu. Next select **ShinyLaser Gcode for BoXZY** from the menu, indicated with the red arrows
- The **ShinyLaser Gcode for BoXZY** window will open and you will enter a filename that ends with .gcode, as indicated by the blue square. Select apply and the file will be saved

Step 5 — Load ShinyLaser Gcode for BoXZY into Repeater-Host

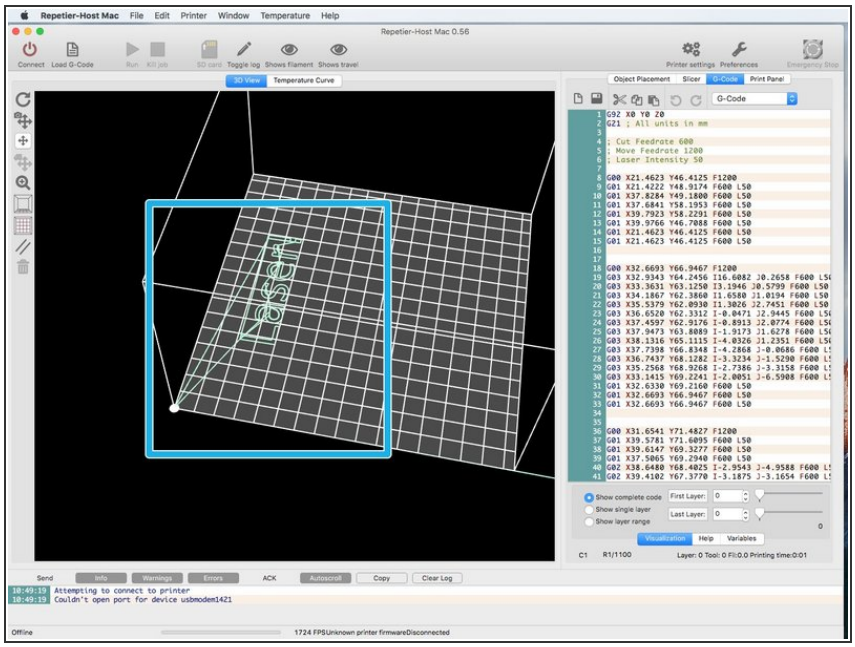






- Open **Repetier -Host**, select **Load G-Code**, as indicated by the red square in the image. Locate the file you saved from the last step and select **Open** as indicated by the blue arrow'.
- Select the Gcode tab in your Repeater-Host interface and examine your Gcode, as indicated by the red square. **Notice** how the file we have loaded in the image is not positioned in the build area correctly, as indicated with the blue square.
- You will not be able to reposition the file in Repeater-Host so you will need to reposition your vector(s) as seen in the image, indicated with a orange square. Additionally you can change your document size settings to *165mm x 165mm* to match the BoXZY Build platform, if desired.
- Save your new file and load it into Repetier-Host

Step 6 — Focusing the Laser



- With you new .gcode file loaded you will see that the vector is in the build area, as indicated by the blue square. Adjust the location of the vector in Inkscape to position the vector where you would like it.
- With the laser head removed, home your **Z Axis**. Measure the thickness of the material that you are going to laser. Add that measurement (in millimeters) to laser focus distance of **50.7mm.**
- Move your Z Axis down(positive) manually in the Repetier-Host controls or with the Gcode sender to the *combined value you just calculated* .
- Now type **G92 Z0** in the **Gcode sender** line and select send to set your Z0 location. Your laser will now be in focus.
- Place your laser in the mount making sure your Laser is secure and connected correctly, that your material is positioned how you want it and that you have your laser safety glasses. Press **Run** to run your file

Thanks again to Walter for the plugin and to Ken Gross for his Pronterface BoXZY hack he posted in the BoXZY community guides!