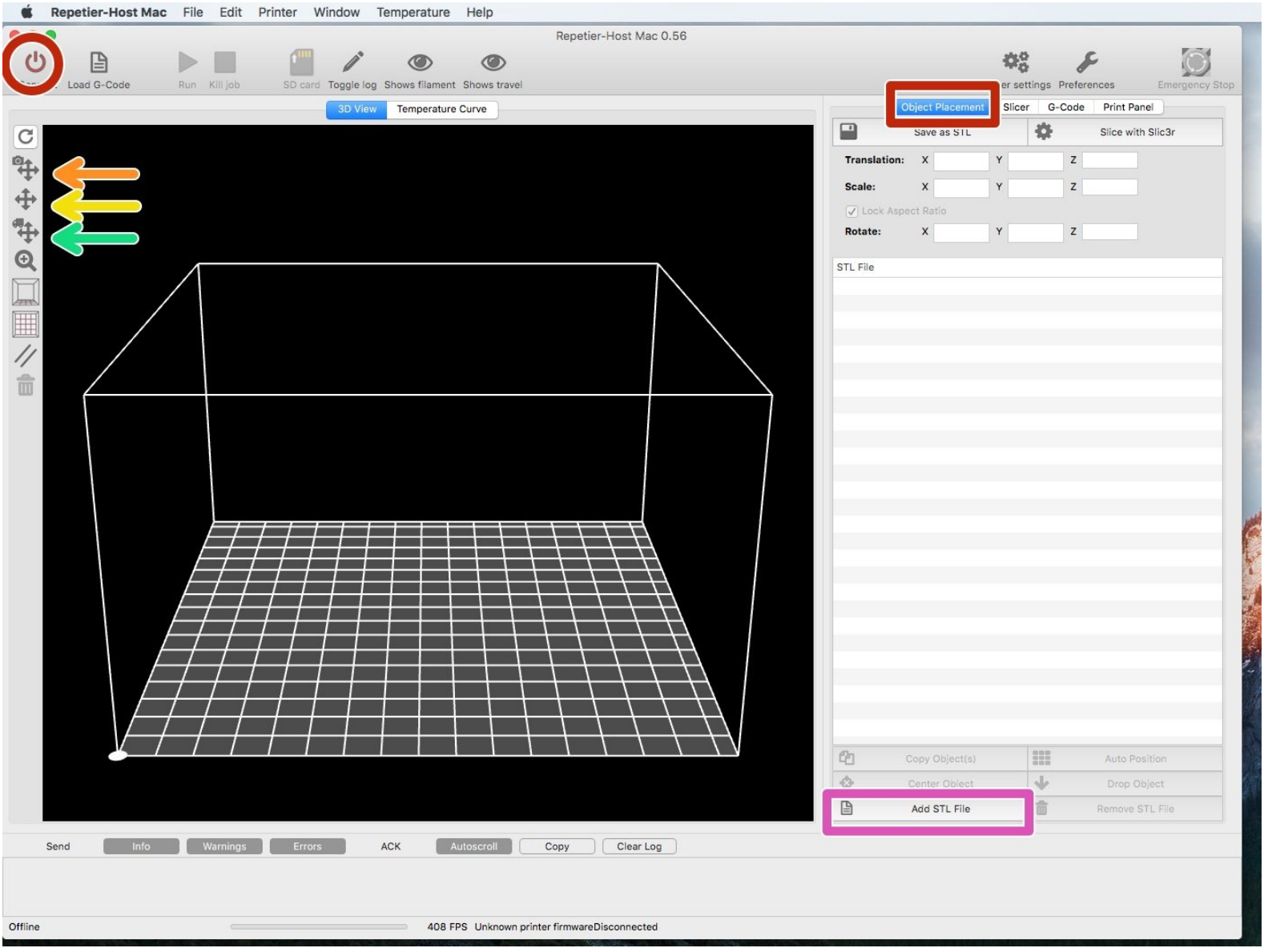
Examining the Repetier-Host interface on Mac OS X

This guide will get you familiar with the Repetier-Host interface on Mac OS X, it looks different than the Windows and Linux interfaces

Written By: BoXZY



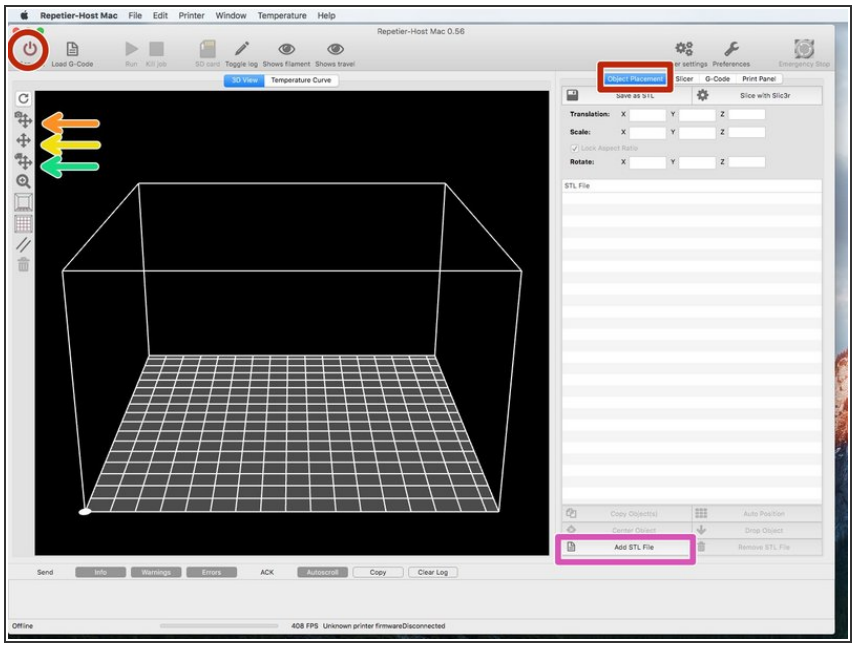
This document was generated on 2020-01-28 12:33:13 PM (MST).

INTRODUCTION

This guide will get you familiar with the Repetier-Host interface, complimentary to our other guides

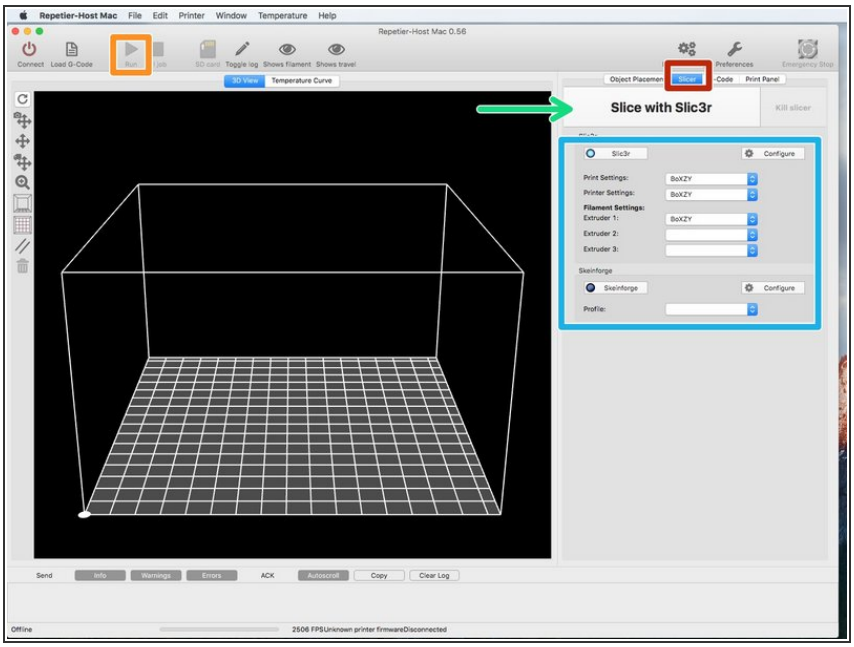
© 2020

Step 1 — Object Placement



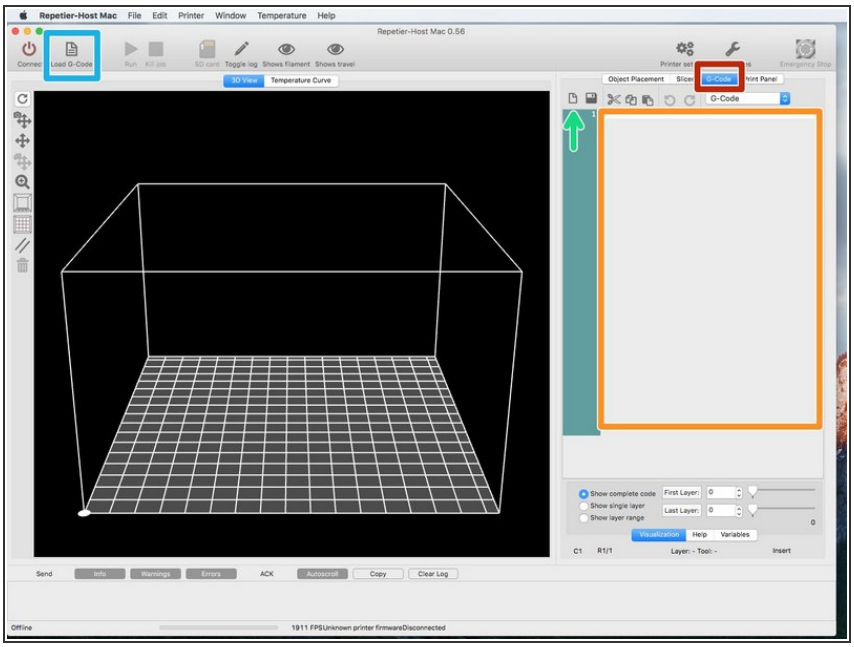
- Open Repetier-Host and select the **Object Placement** tab, as indicated by the red square in the image
- Connect to your BoXZY with the USB cable and select **Connect** as indicated by the red circle in the image
- Load an *.stl* file for 3D printing by selecting **Add STL File**, as indicated by the pink square in the image
- Move the preview viewing angle by selecting **Camera Move** and moving your cursor in the preview, as indicated by the orange arrow in the image
- Move the build area in the preview by selecting **Move** and moving your cursor in the preview, as indicated by the yellow arrow in the image
- Move an object from a loaded STL file by selecting **Move Object** and moving your cursor in the preview, as indicated by the green arrow in the image

Step 2 — Slicer



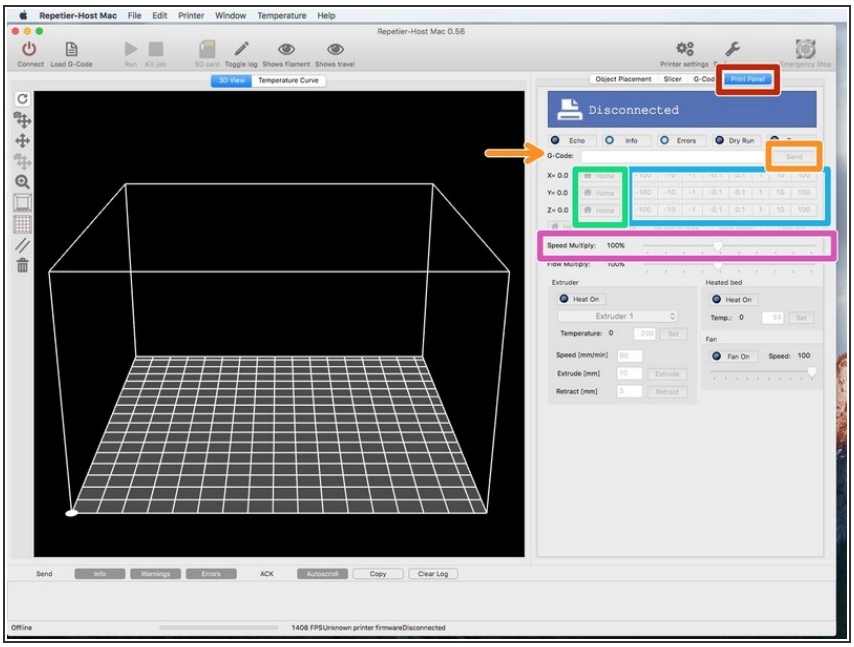
- In Repetier-Host select the **Slicer** tab, as indicated by the red square in the image
- Verify that your **Print Settings, Printer Settings and Filament Extruder 1** settings menu's have the BoXZY configuration preferences selected, as indicated by the blue square in the image
- If they are *not* available in your menus you have not saved BoXZY settings in the configuration of Slic3r. Please follow our Interface Solution for Mac OS X and BoXZY Guide, and follow Steps 4 through 8 to get you set up for your first print
- With your BoXZY settings loaded, your STL file loaded select the **Slice with Slic3r** to prepare you file for printing, as indicated by the green arrow in the image
- With your BoXZY 3D printing platform leveled and zeroed select Run, as indicated by the orange square in the image, and your BoXZY print will begin

Step 3 — G-Code



- In your Repeater-Host interface select the **G-Code** tab, as indicated by the red square in the image
- Load a G-Code file by selecting **Load G-Code** as indicated by the blue square in the image
- Once you load the G-Code you will see it in the G-Code window, as indicated by the orange square in the image
- One was to prepare G-Code is with Autodesk Fusion 360, which is provided with every BoXZY
- When you are finished milling or want to load a new file select **New File**, as indicated by the green arrow in the image, and your G-Code will be removed from the G-Code window

Step 4 — Print Panel



- In your Repeater-Host interface select the **Print Panel**, tab as indicated by the red square in the image
- In your Print Panel you will see your **limit switch home buttons** as indicated by the green square in the image
- In your Print Panel you will see buttons to move your Rapid Change Attachment incrementally from 0.1 to 100mm positively and negatively, as indicated by the blue square in the image
- In your Print Panel you will see the **Speed Multiplier** sliding bar to adjust the movement of the Rapid Change Attachment, as indicated by the pink square in the image
- In your Print Panel you will see a **G-Code Sender** code line and **Send** button, as indicated by the orange arrow and square in the image
- Note that in our guides there will be G-Code that you need to enter, use the G-Code Sender to execute those G-Code commands. One example is *G92 X0 Y0 Z0* which will assign the current location of the X, Y and Z axis at 0,0,0

© 2020