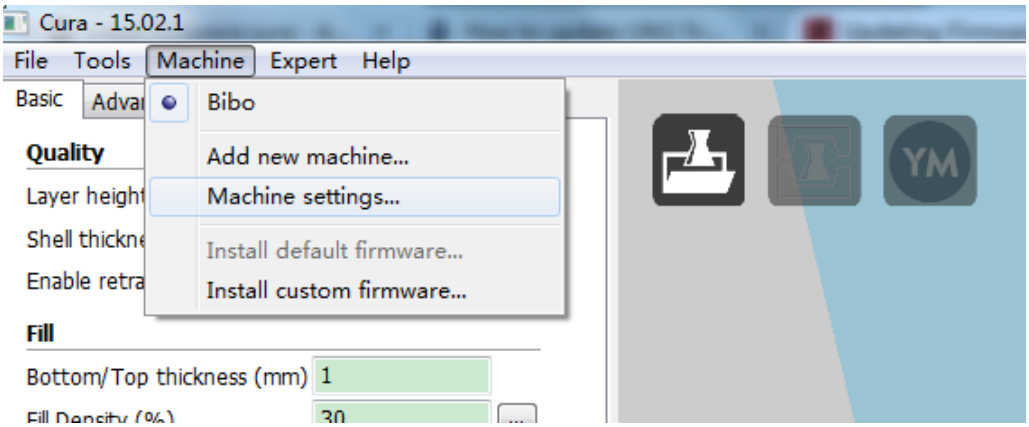
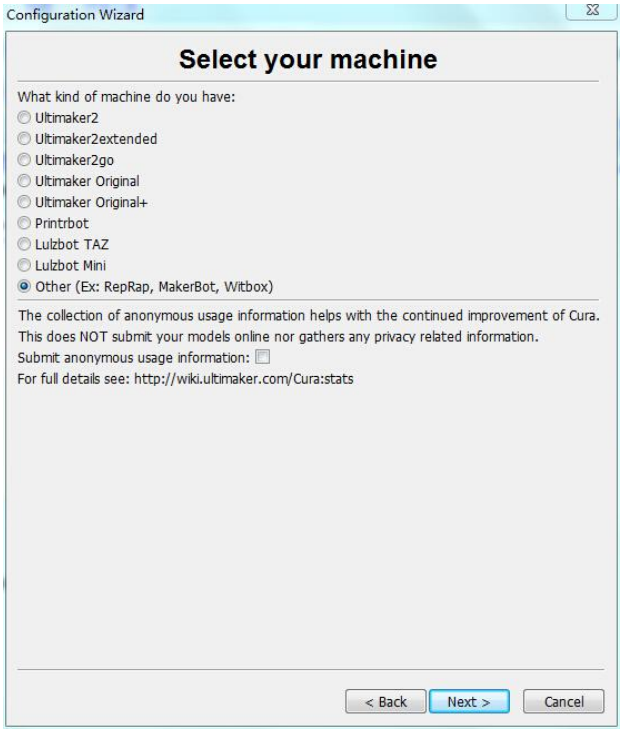
How to use BIBO cura settings in Cura software for maker touch model

1. **Copy the cura installer from SD card's BIBO touch maker cura settings folder to your computer (if downloading from the internet, the version is 15.04.6).**
2. **Install the cura on your PC. (Mac users can check page 11 in this document for how to install cura on mac. if it is your first time to install the cura, cura will let you choose a machine model or create a new machine such as BIBO 3D. If you have already installed cura, you can add a new machine in cura. Both two ways are almost the same)**
3. **Adding BIBO machine in Cura. Open up Cura. Click "Machine" on the menu bar and click "Add new machine"**

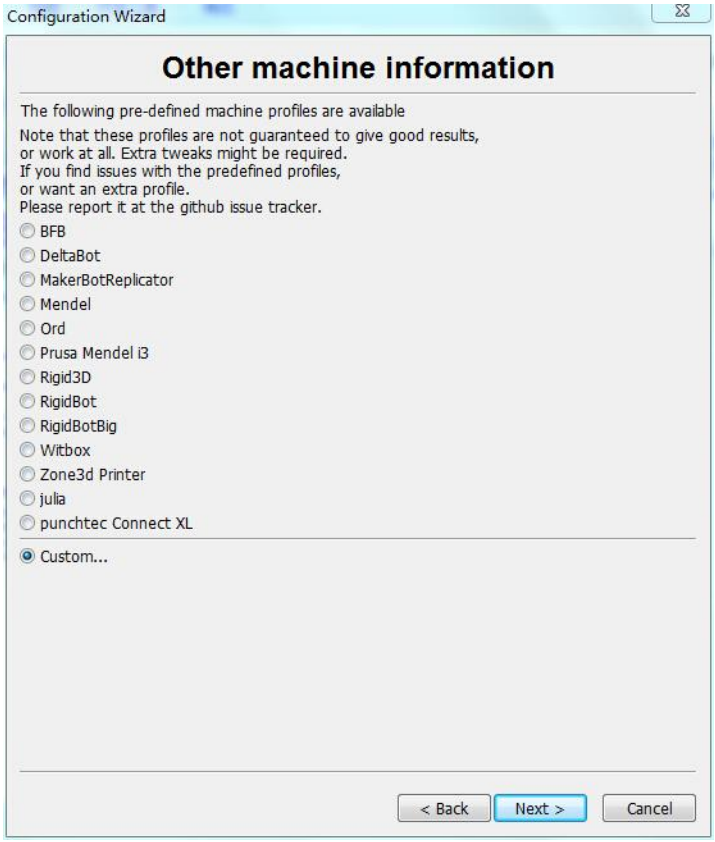


Click "Next",

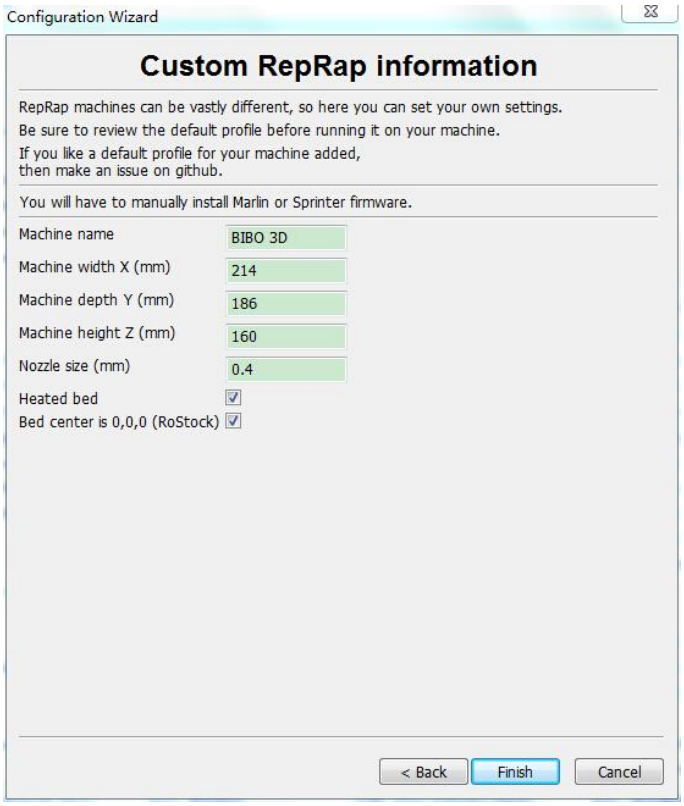
Choose "Other"



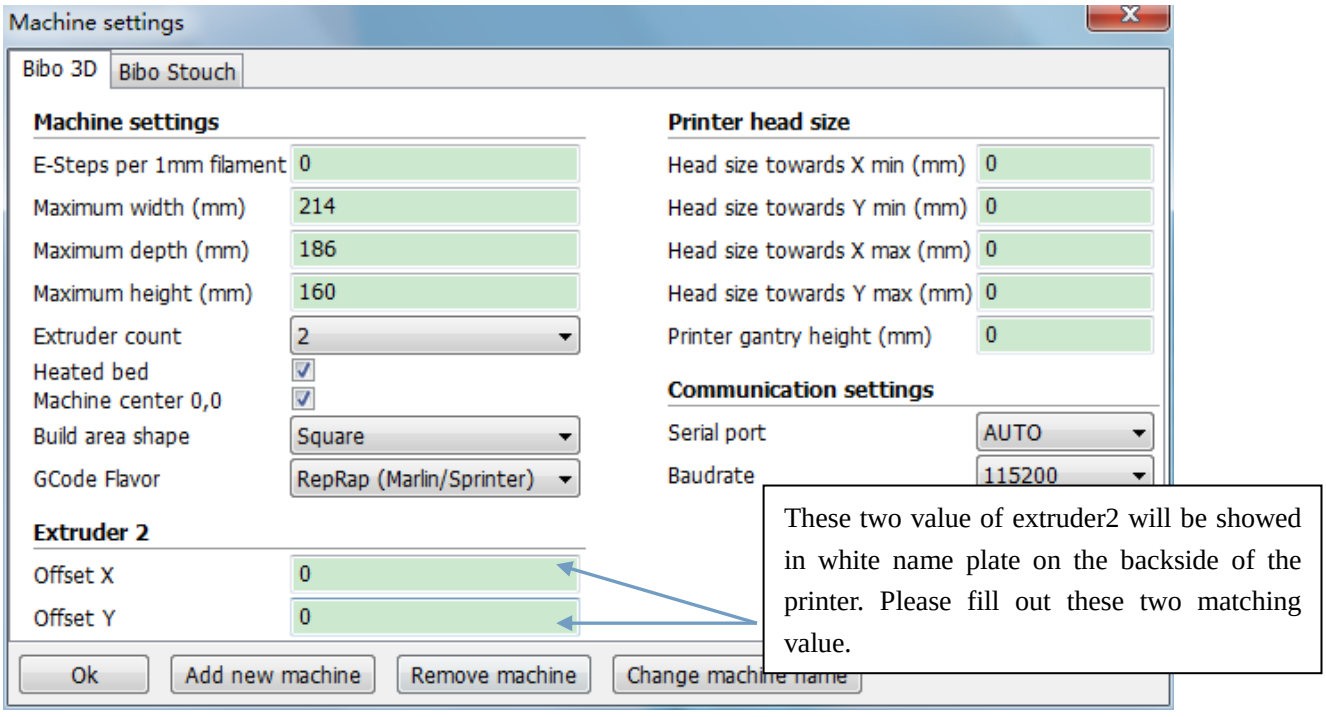
Choose "Custom"



Fill out the machine information as below and click "Finish"

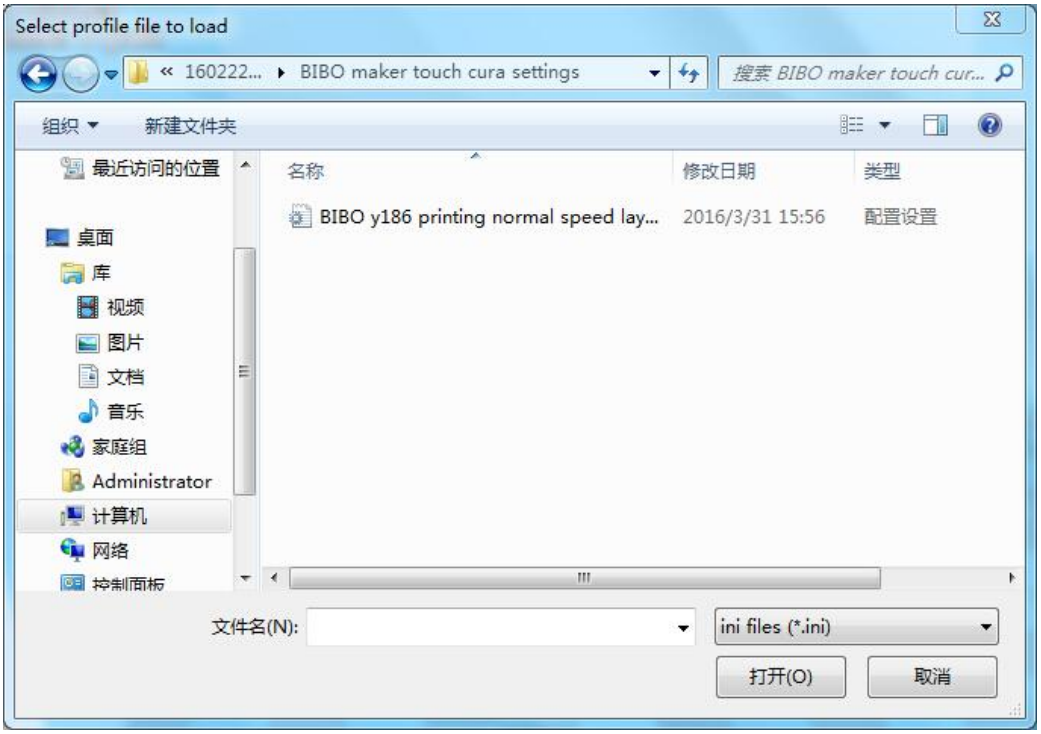


Now BIBO1 machine's configuration is finished. But for BIBO2 model, please click "Machine" on the menu bar and click "Machine settings". Choose your model BIBO 3D and change extruder count to 2. GCode Flavor should be RepRap(Marlin/Sprinter). Click "ok" in the end. Then click "Machine" on the menu bar and click "Machine settings". Choose your model BIBO 3D again, then fill out the offset of extruder 2

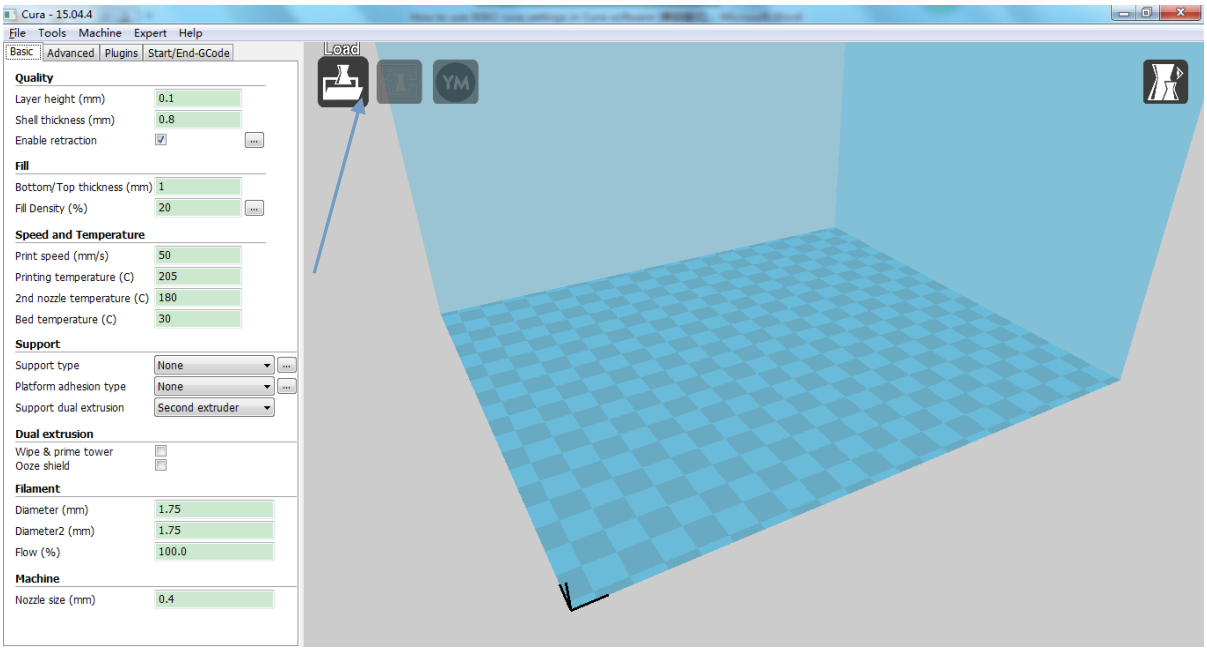


4. After adding the machine, you can go to "File" on the menu bar and click "Open Profile…".

Choose the right ini format file to cura from SD card we sent to you and click "Open", now BIBO cura settings is in the cura and you can slice the object you want.



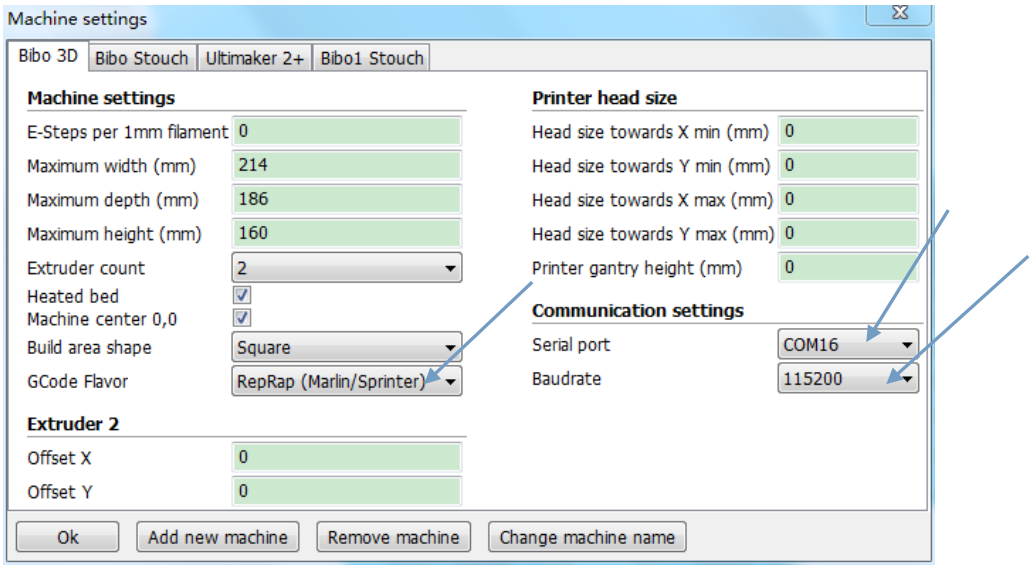
5. **Load the object to cura, and G-code will be generated automatically. You can also revise cura settings such as layer height, print speed, printing temperature and other parameter you want to revise. After revision, G-code will be generated at the same time automatically.**



6. **Then you can go to "File" on the menu bar and click "Save GCode…" to save it on computer or SD card. Now insert SD card with saved G-code into printer and start print from the SD card.**
7. **If you want to print by connecting the computer with USB cable, then go to the following steps(not recommended, printing from SD card is more stable especially for long time printing), you have to install the driver on the computer for the printer. For how to install the driver, please kindly check the**

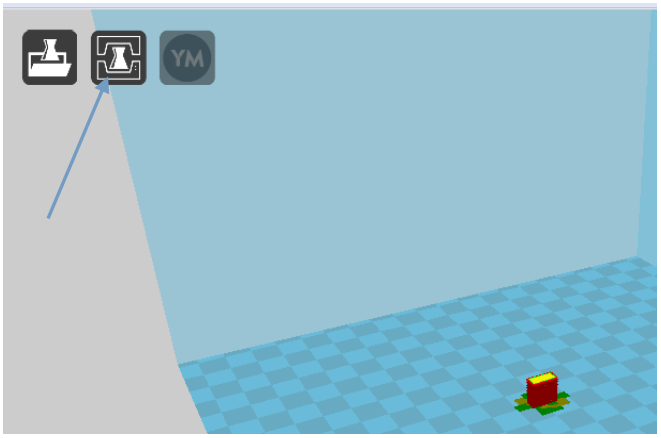
printer operation manual.

8. After driver installation, open Cura(Your printer are still connected to computer by usb cable). please Click “Machine” on the menu bar and click “Machine settings”. Choose your model BIBO 3D and Click Serial port to Choose the COM-port which belongs to your printer, choose Baudrate 115200 and Gcode Flavor should be “RepRap(Marlin/Sprinter)”, and then click ok. Other parameters are not necessary to be revised. If no port shows up, it could because of the following:
   - your printer might not be plugged into the USB port or no power supplied.
   - The driver is not installed, or currently installing by windows.



NOTE: what COM-port is present for your printer after installation. If there are more than one COM ports available, unplug the USB cable of the printer and then re-plug it in again. Check what port number is appearing and disappearing. This port number is the port present.

9. After loading the object or gcode files to Cura, please click the button as the photo below to start print with USB.



SLICING – CURA – 2 Color Printing (print 2 parts, each a different color)

Description:

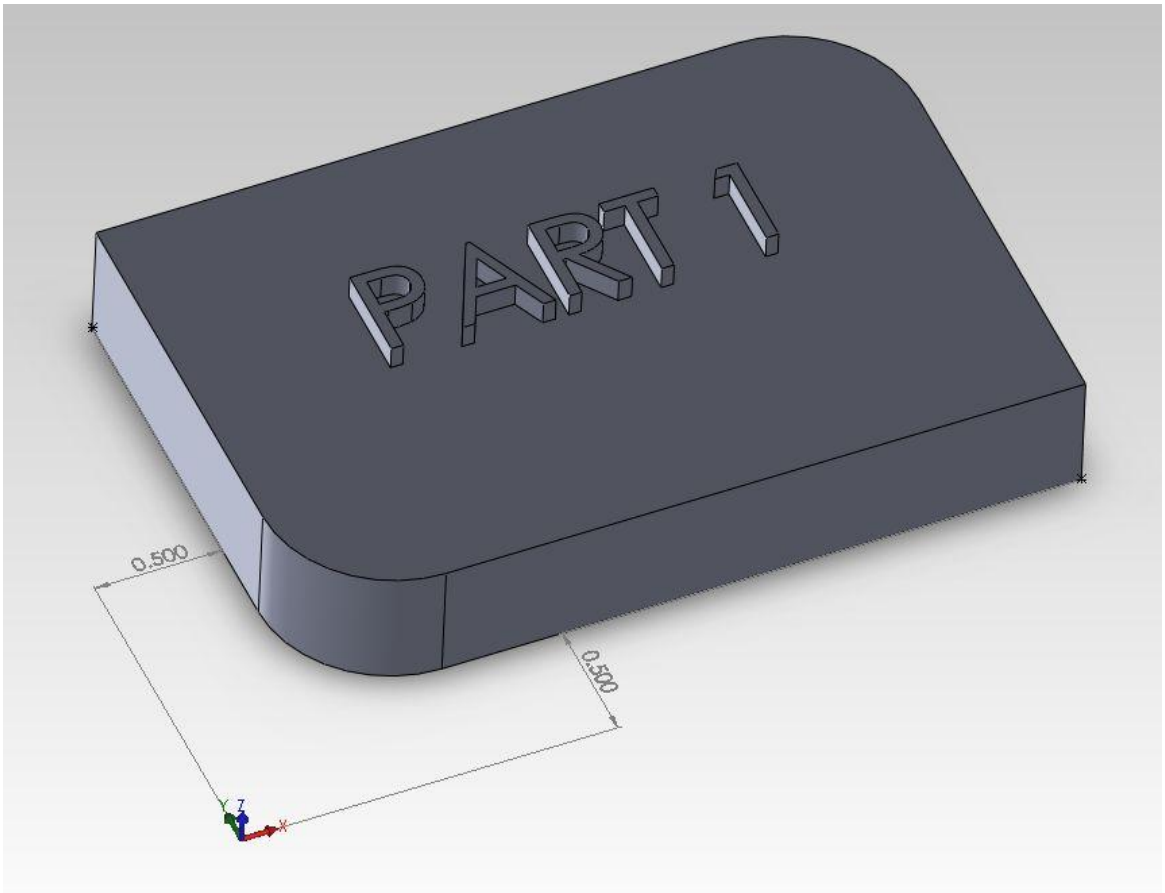
This solution will help you print 2 different parts, each in a different color, at the same time. The process described below is how to make two separate parts that are two different colors.

Process:

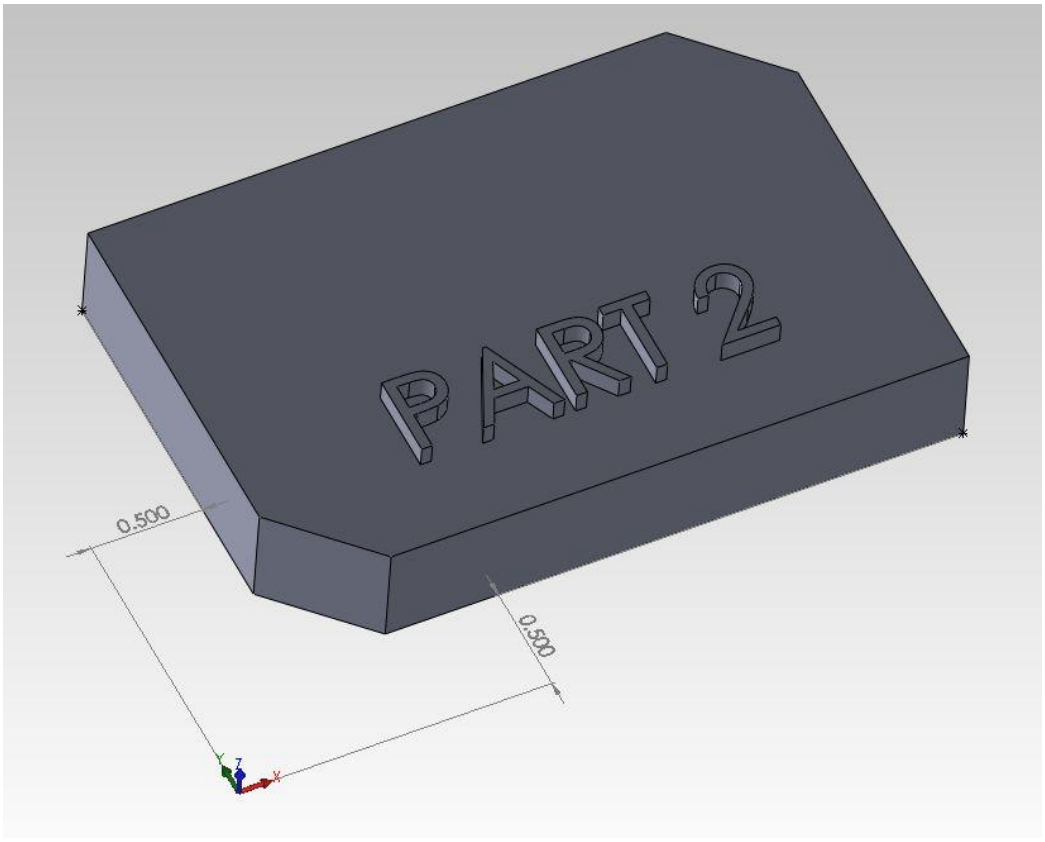
The key to printing two separate parts in dual head mode, is their relation to the origin. To do this properly, the parts to be printed will need to be modified in the CAD software. Without any modification, the parts will likely overlap each other when the “Dual Extrusion Merge” command is executed.

The following is an example of parts that are not aligned properly to the origin:

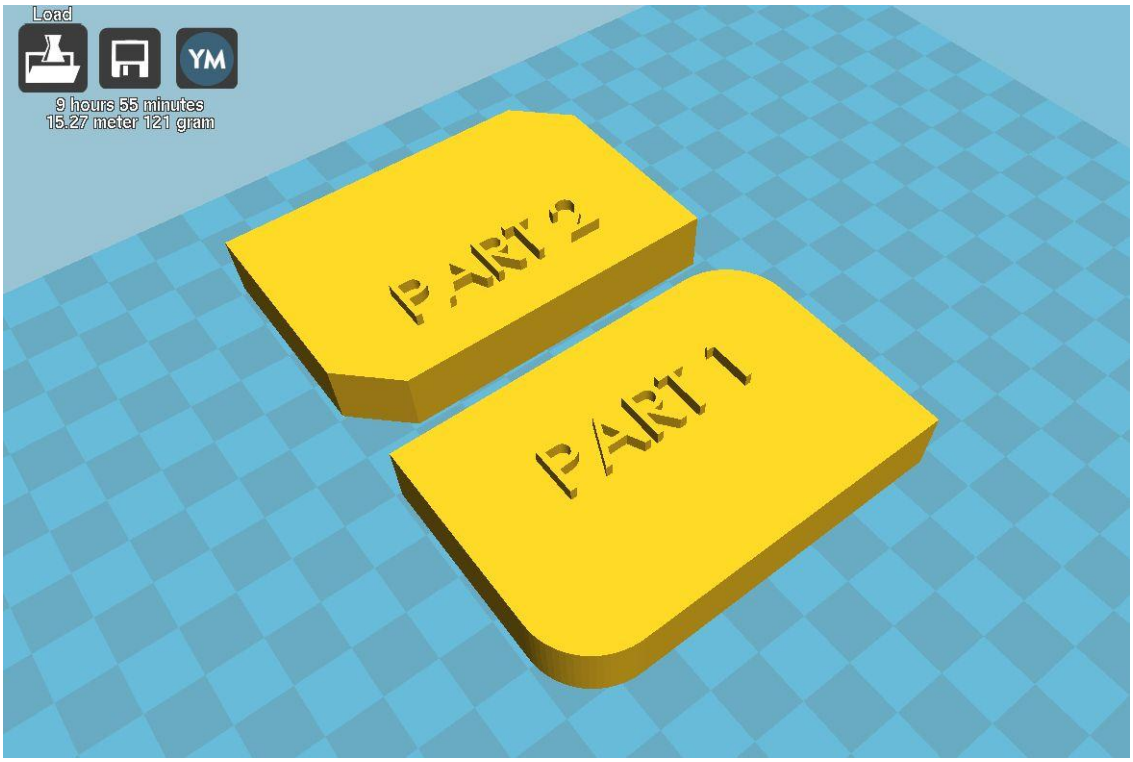
In the CAD software, **PART 1** positioned 1/2″ from the origin in the X and Y directions.



PART 2 was designed separately, but it too is positioned 0.5″ from the origin in the X and Y directions.



When imported into Cura, the two files will be designated as printing with Extruder 1 (yellow).

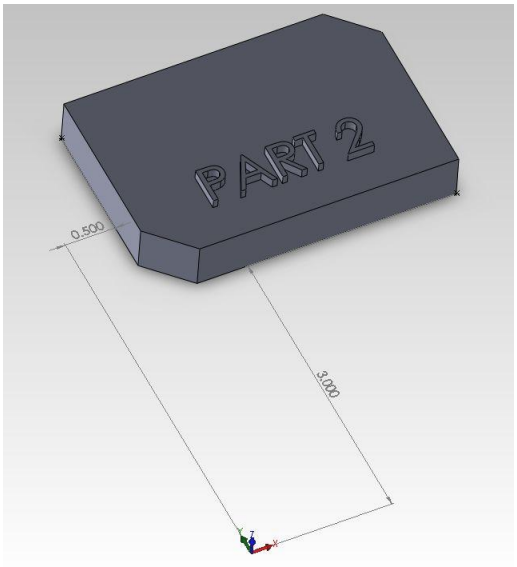


To print with 2 colors, the “Dual Extrusion Merge” command is applied.
Right click a part and select [Dual extrusion merge].
However, since the parts are positioned in the exact same spot relative to the origin, they overlap.

This is not the desired result as this part will be printed merged as one piece.Extruder 1 – YELLOW, Extruder 2 – RED

In order to prevent this undesired merge, one of the two cad files will need to move relative to the origin.

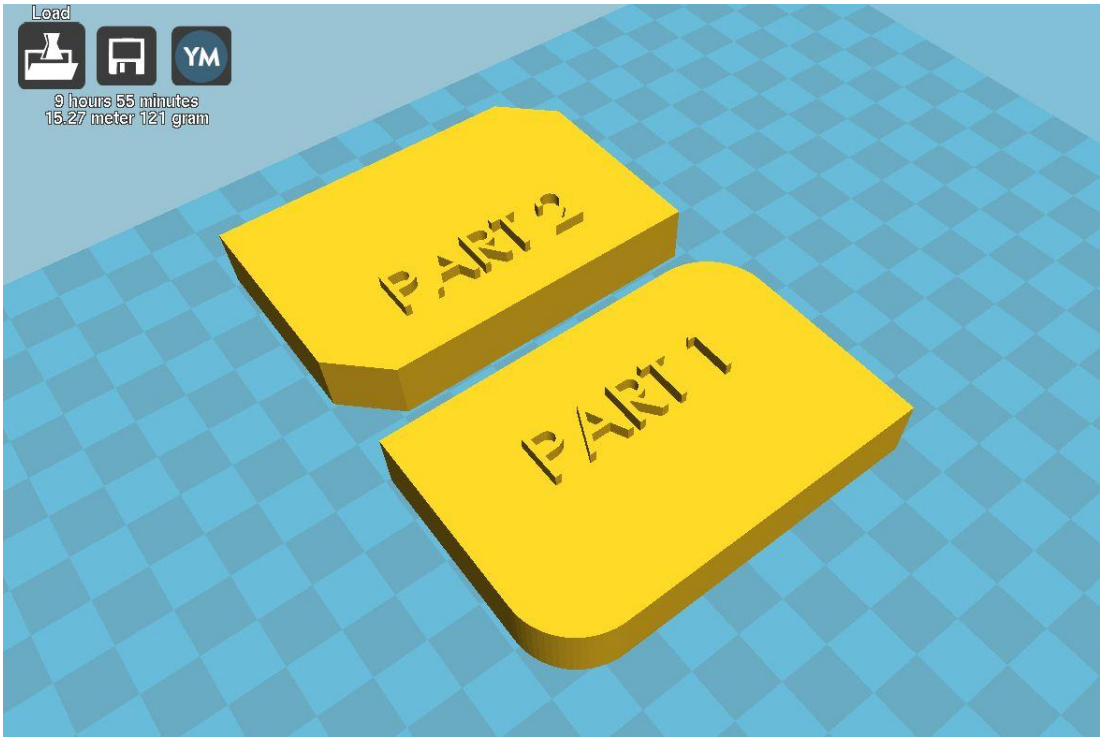
PART 1 will remain the same but **PART 2** will move 2.5″ in the Y direction to accommodate the area **PART 1** will occupy.



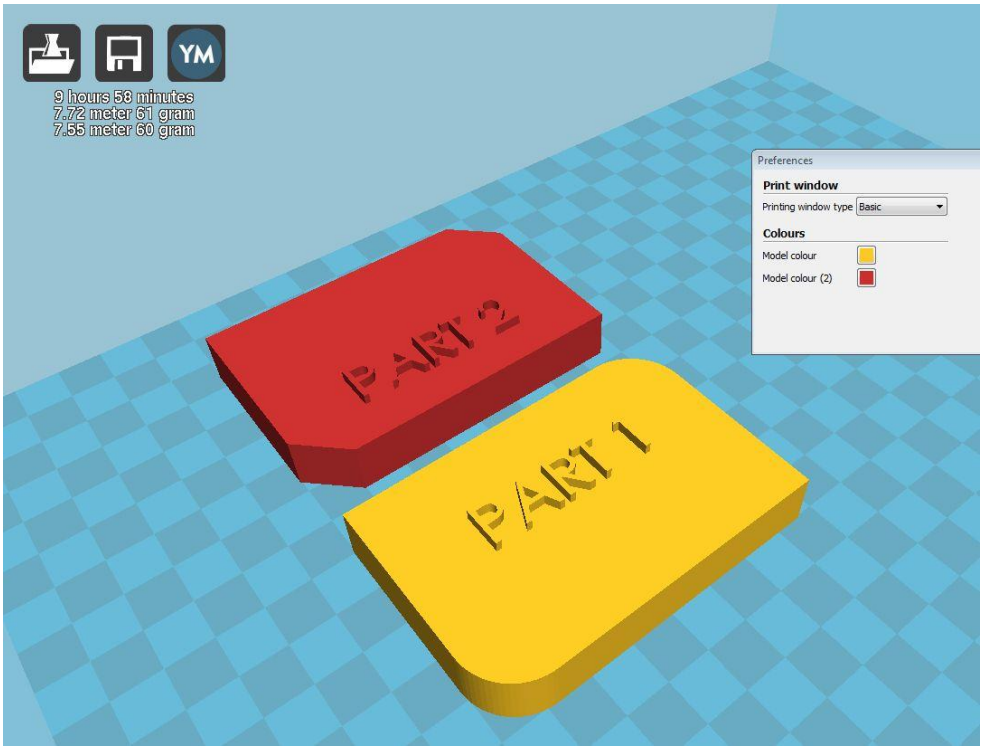
This shift will now set **PART 2** 1/2″ away from **PART 1** with the “Dual Extrusion Merge” command.

The new **PART 2** is saved as an new STL and both parts are reloaded into Cura.

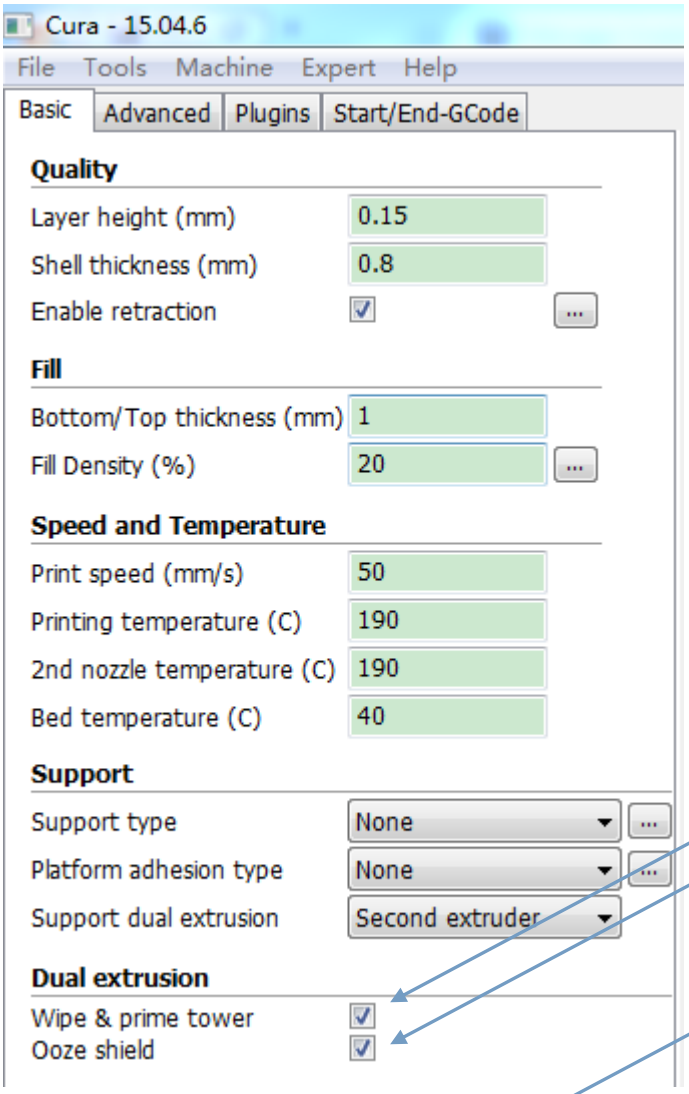
Again, both parts are registered as being printed with Extruder 1 (yellow).



Now, execute the “Dual Extrusion Merge” command ant the parts will no longer be overlapping. This is because the origin for **PART 2** is now 2.5″ further from the part in the Y direction.



Now your two parts are ready to be printed in two separate colors.
These parts can also be multiplied with the “Multiply Object” command.
After objects placed in cura, the settings for cura should be also changed, you should choose **wipe & prime tower and Ooze shield** in basic tab as the photo below:



This is the Ooze shield. It can catches any oozing from the unused nozzle in dual extrusion. **Please take this shield away after printing.**

Now you can save the gcode in computer and copy it to SD card for printing.

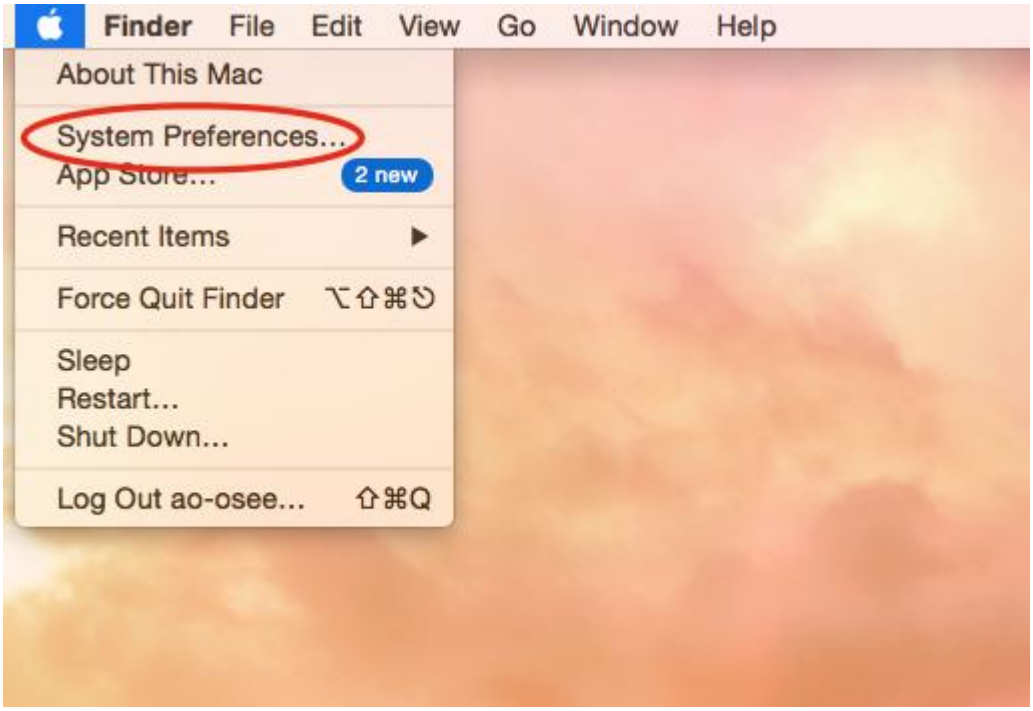
How to install cura for mac users

You'll need to allow app installation from sources outside of the Mac App Store. Follow the instructions below based on your version of Mac OS X.

Mac OS X 10.10 (and later)

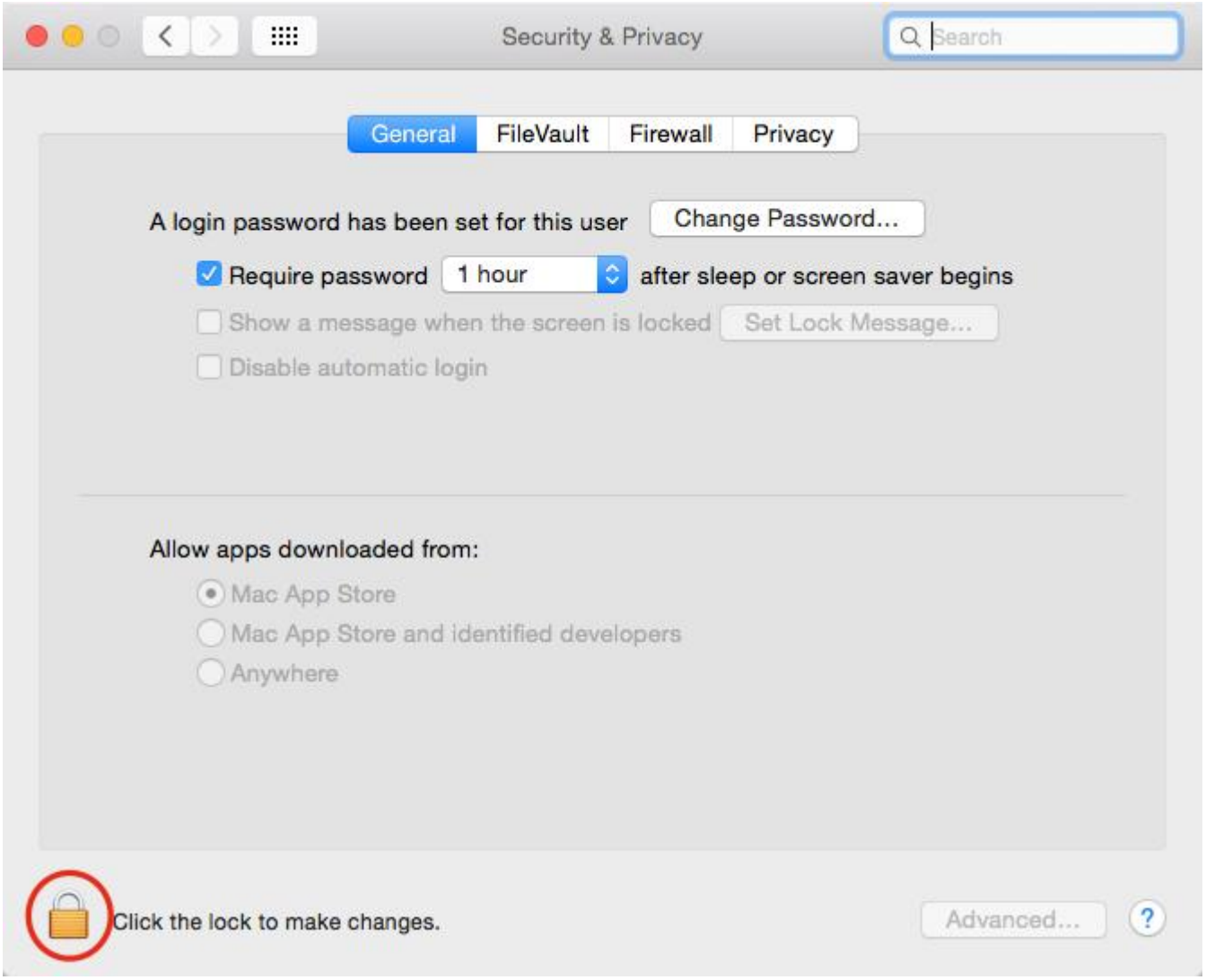
Click on the Apple icon in the upper left corner of the screen.

Select **System Preferences**

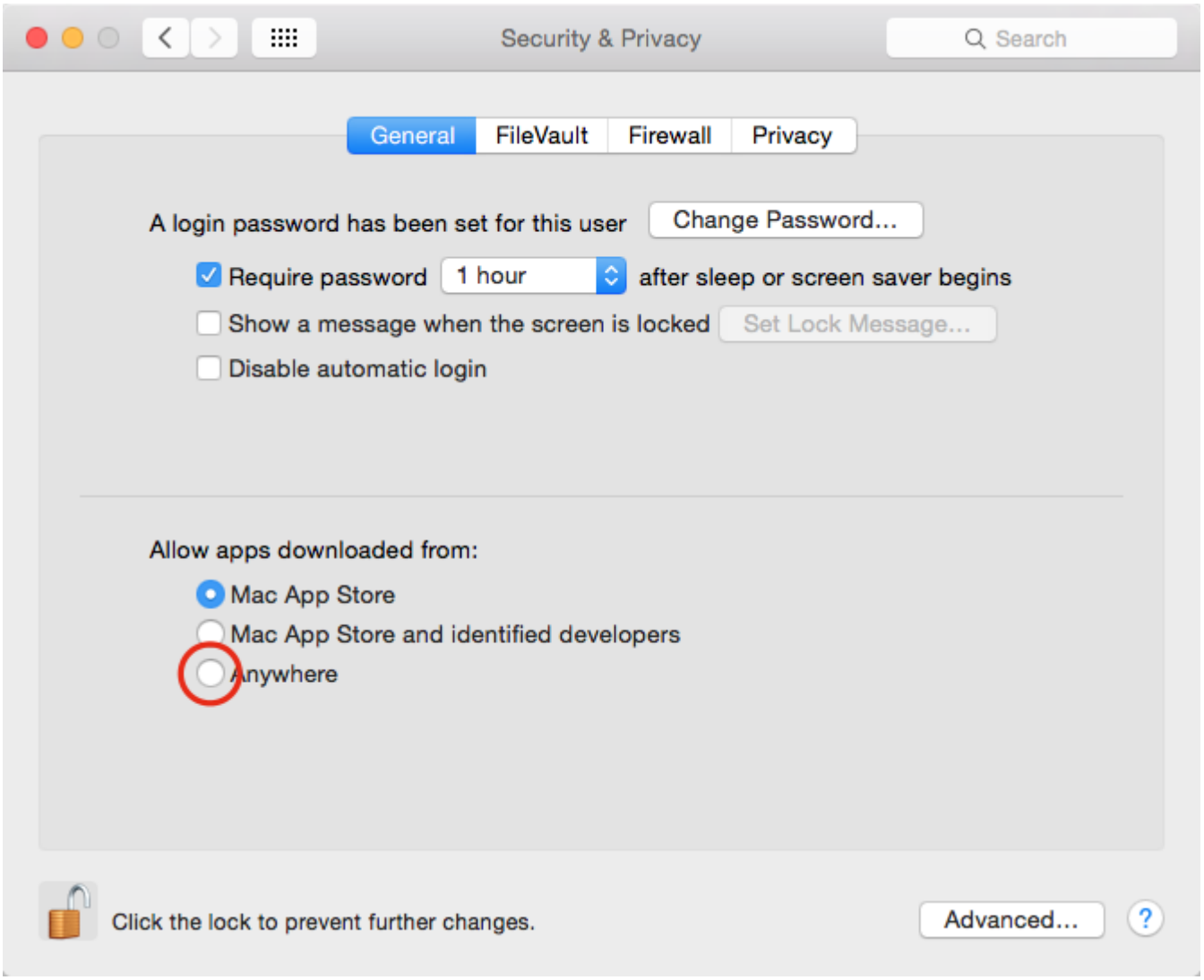


Select **Security & Privacy**

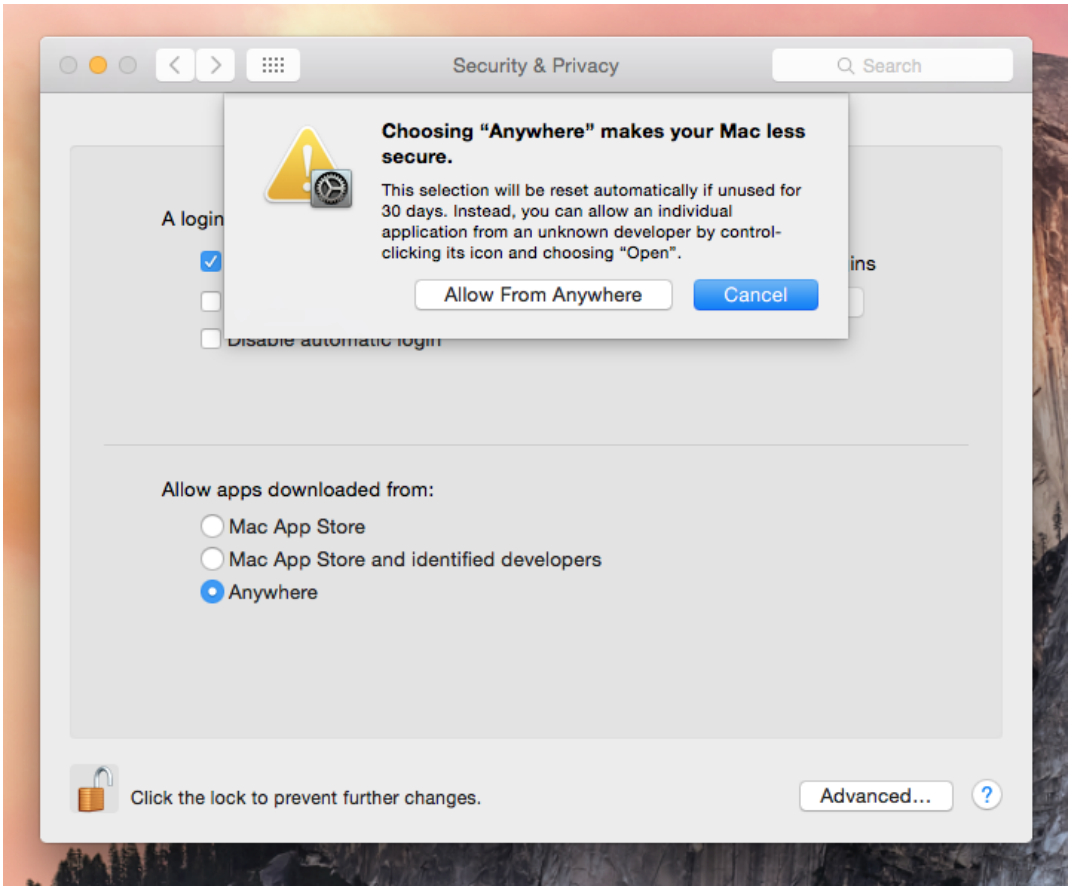
In the **Security & Privacy** window click on the lock icon in the lower left hand corner of the screen.



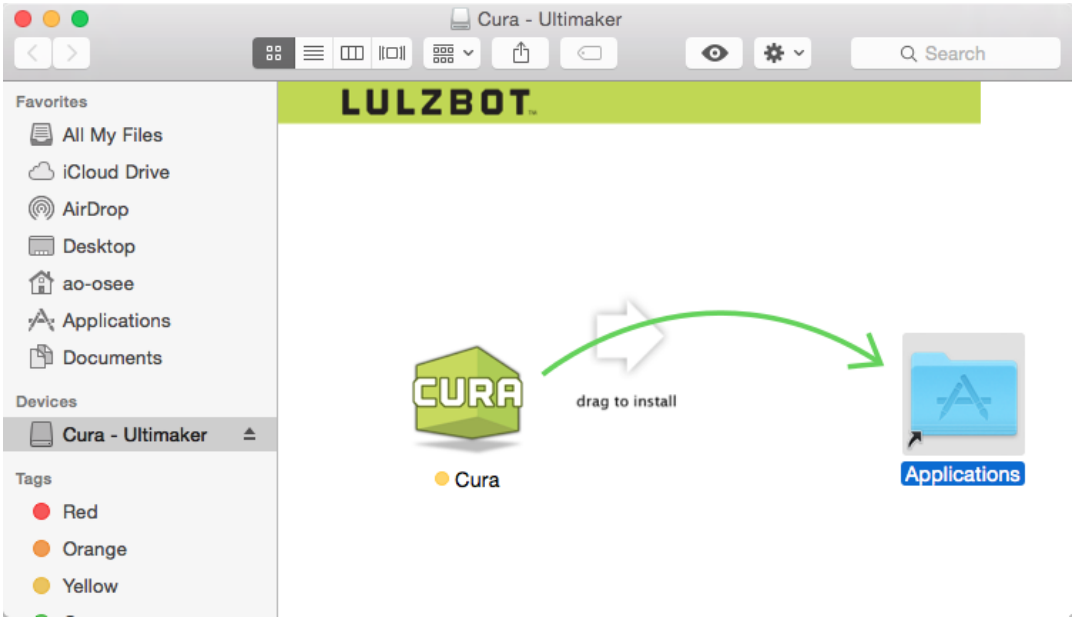
In the section titled: **Allow apps downloaded from:** select **Anywhere**.



In the next pop-up window select **Allow From Anywhere** to continue.

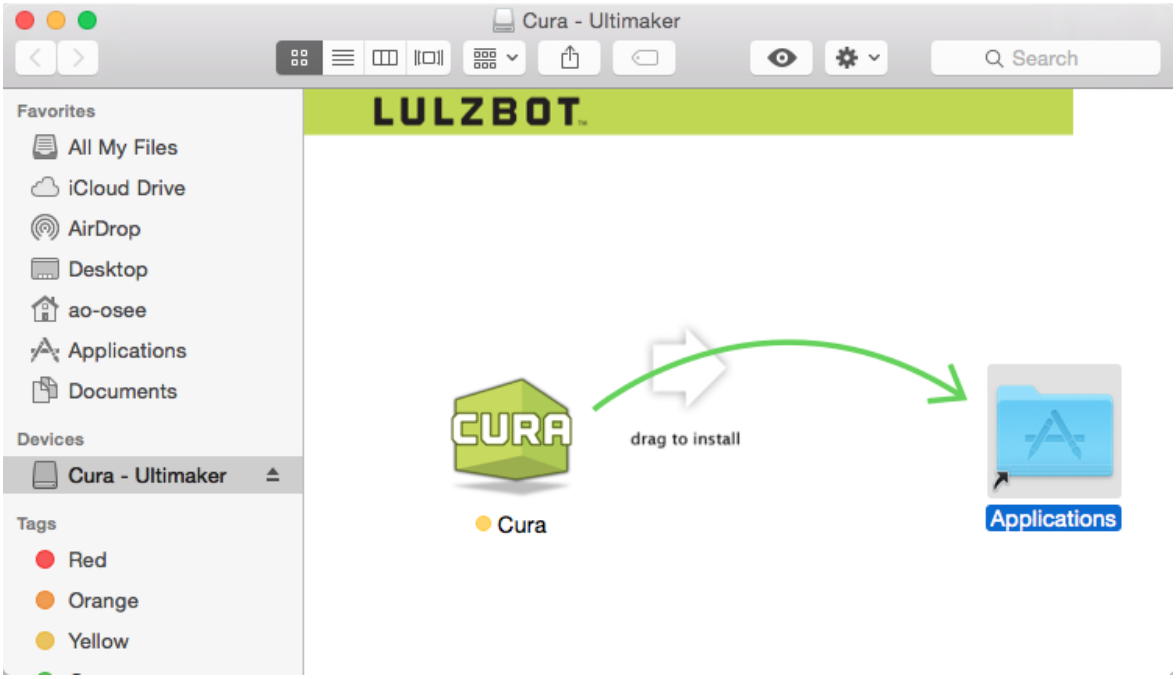


Open the folder containing the file copied earlier.
Drag the Cura app file into your **Applications** folder.



Click on the **Cura** icon to open Cura!

Mac OSX versions earlier than 10.10
Open the folder containing the file copied earlier
Drag the Cura app file into your **Applications** folder.



Click on the **Cura** icon to open Cura