Bucknell UNIVERSITY

Anycubic Photon Mono X

Basic tutorial for use of the Anycubic resin printer

Written By: Jacob Nickle

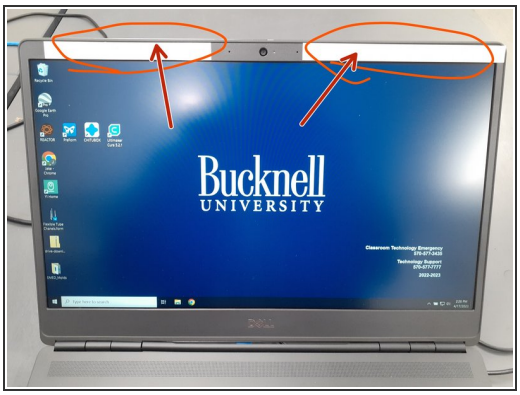


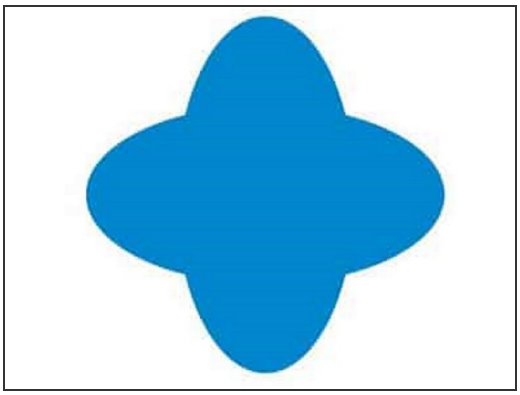
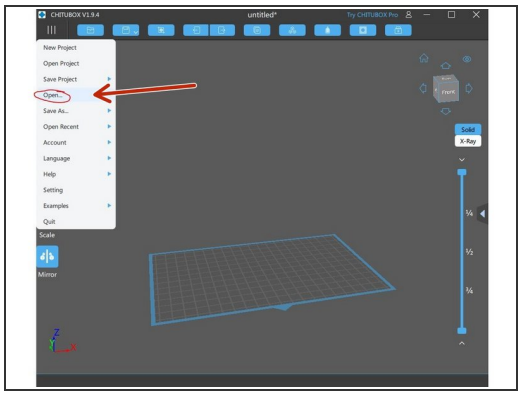
Step 1 — Anycubic Photon Mono X



- Anycubic Mono X is a LED based SLA resin printer. Unlike the FormLabs, each layer is cured all at once using an LED screen to cure each layer
- When compared to FDM printing, SLA printing has better resolution and less noticeable layer lines. Ask the Lab Director to know if resin printer is best used in your project
- This tutorial will teach only the basics of resin printing to get you started and is not an all-inclusive guide to resin printing
- It is assumed that the user has a 3D stl model that fits within the build volume of the printer

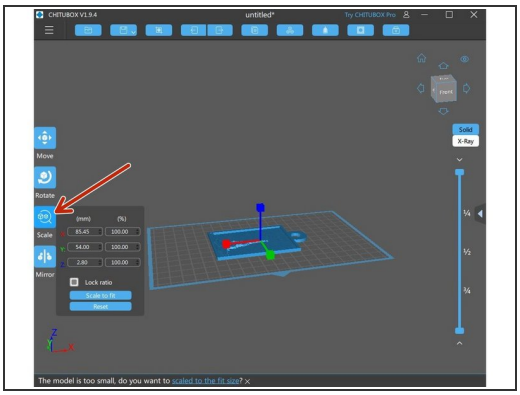
Step 2

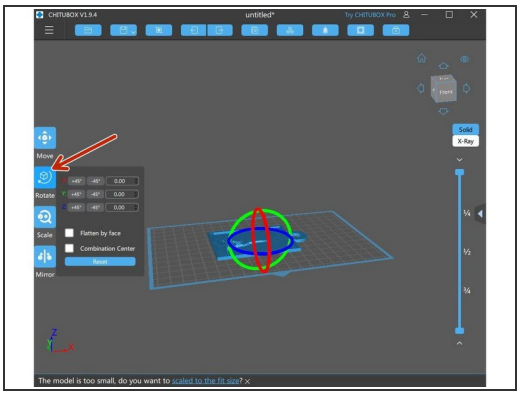


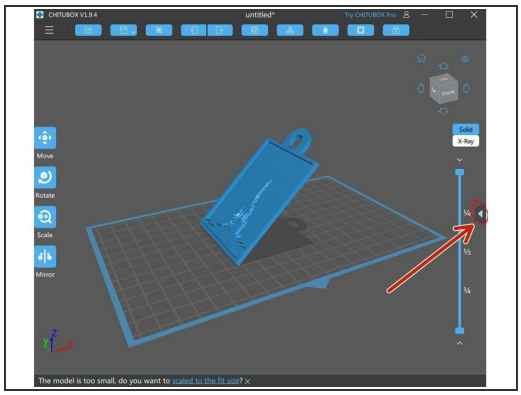



- Log onto the BME computer in the Fab Lab using credentials provided at the top of the screen
- Open the Chitubox slicer software
- Import stl file into Chitubox

Step 3



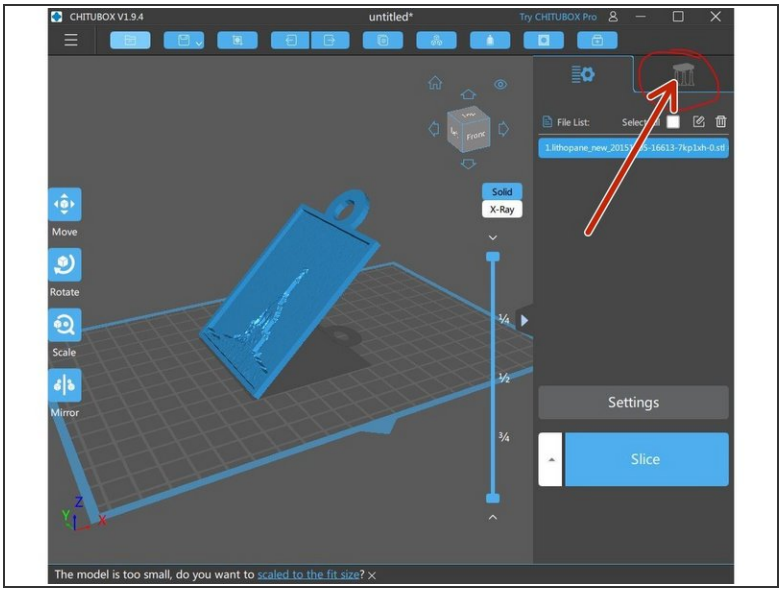


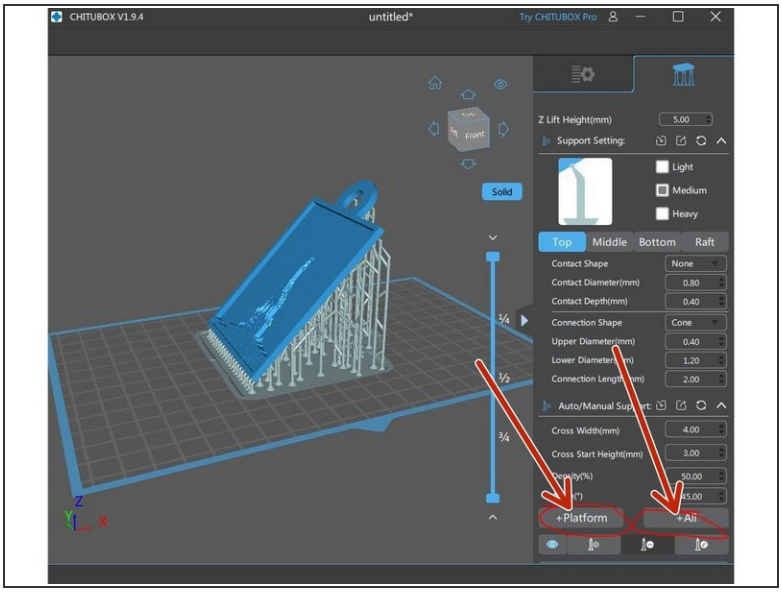


ⓘ Further information than given in this tutorial can be found here.

- Roughly verify scaling of part by visually identifying size of object on the screen. If needed, scale your part
- Rotate your part as needed. Note that you are able to select a surface that you want on the build platform.
- Open further model settings using arrow in picture(right center of screen) if not already open

Step 4





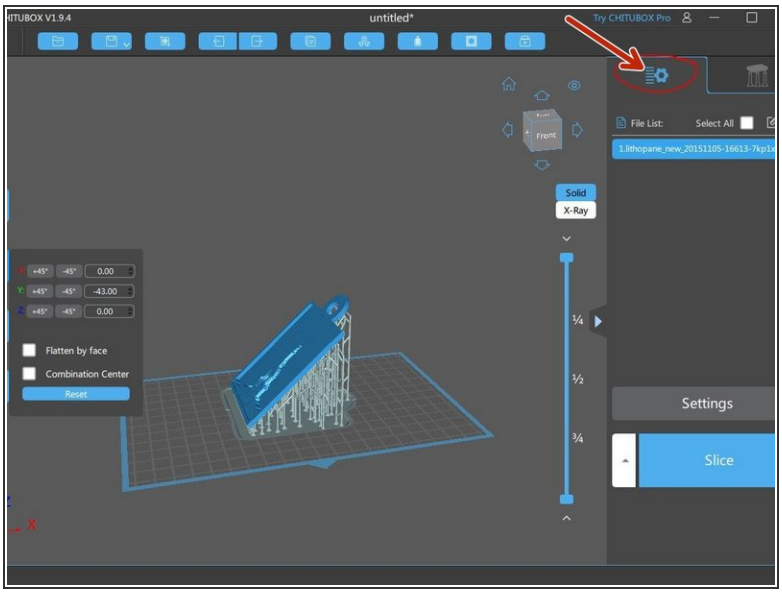
- open up support material / platform generation tab

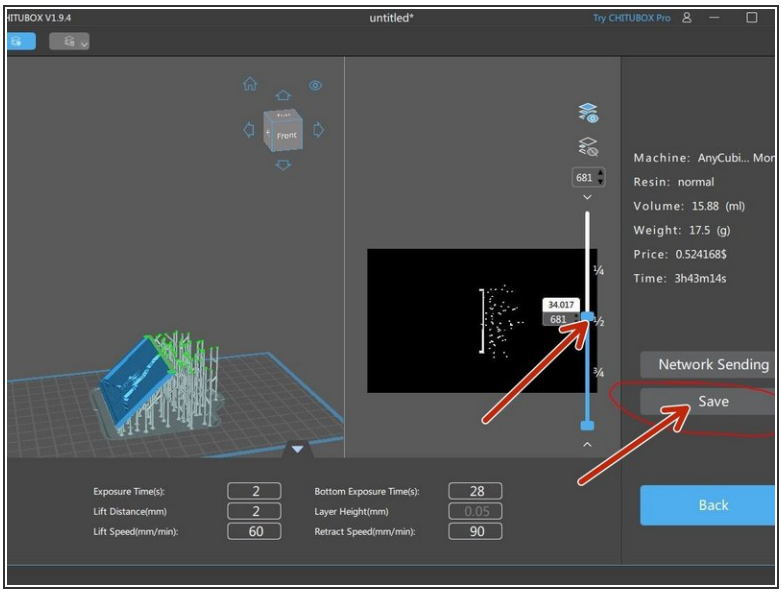
ⓘ Support material is used to support your model in areas that would otherwise not stick to the build plate

ⓘ Platforms are used so that support material can stick to the built plate more effectively and so that the model can be removed more easily

- Autogenerate supports and platform using the +Platform/+All buttons

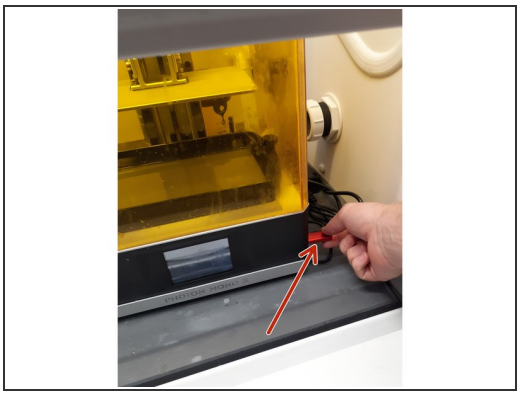
Step 5



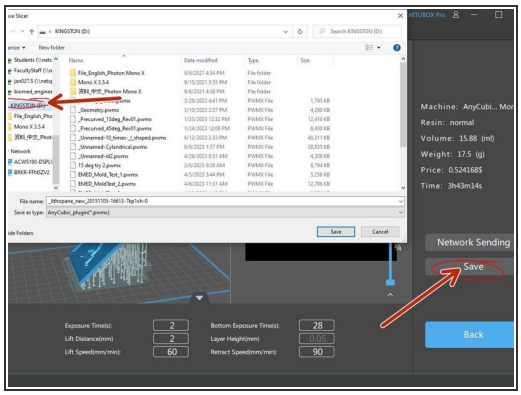


- Once you are satisfied with your print, return to the main view by selecting the left tab
- Slice print by selecting the blue "Slice" button
- Verify that the print looks as you expect it to

Step 6

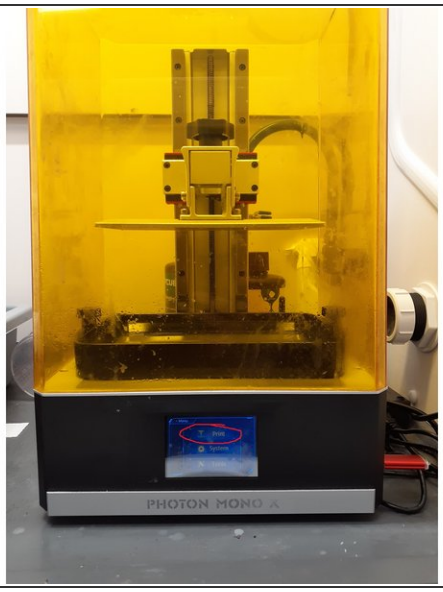






- Take USB from the Anycubic resin printer
- Insert USB in Fab Lab computer
- Select "Save"
- Save file in USB
- Safely eject USB using the "^" on the bottom right of screen, right clicking USB icon and selecting eject.
- Remove USB from computer and put back into the Anycubic resin printer

Step 7

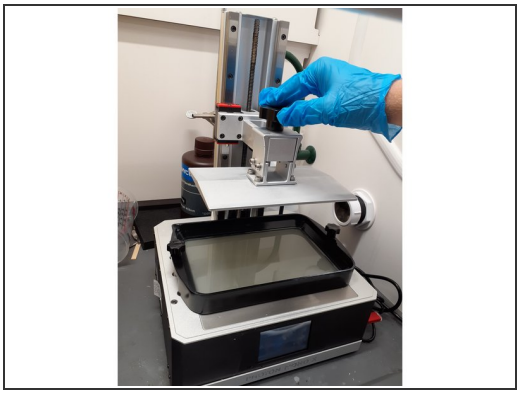




- Verify build plate is secure and there is a sufficient quantity of resin
- If you need to put more resin in, you can use the Phrozen ABS-like resin. If you have questions or need to change to a different type, please ask the Lab Director
- To start the print, press start, select file and press play

Step 8

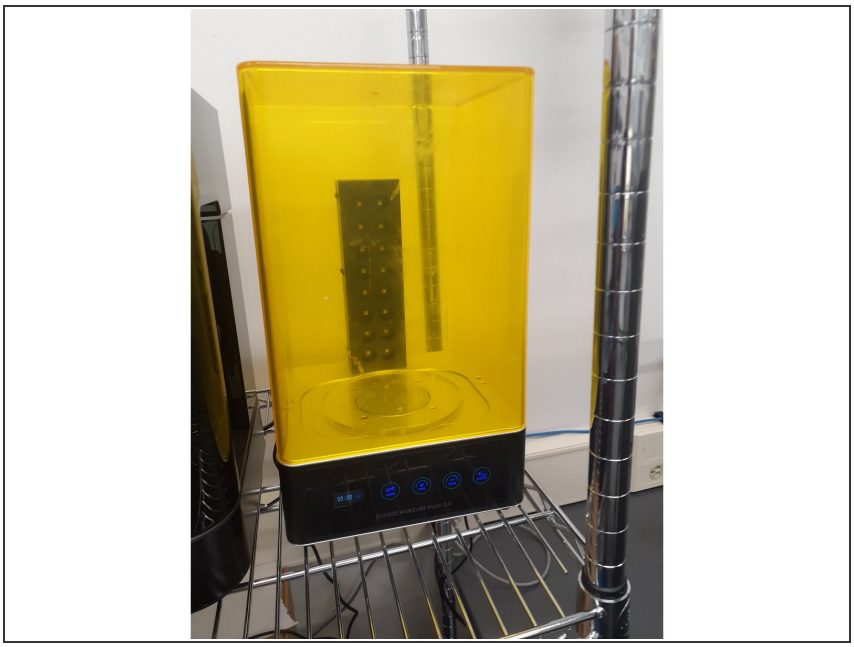






- Once the print is complete, the next step is to put on gloves
- Open the Formlabs isopropyl alcohol wash station **closest** to the Formlabs resin printer
- Remove transparent cover on the Anycubic printer
- Grab a paper towel to prevent resin from dripping anywhere during the next two steps
- Loosen build plate as shown by unscrewing hand screw
- Place build plate within basket such that it will not scrape the sides of the wash station when closing
- Wash for 20 minutes

Step 9



- After washing the print, again put on gloves
- Open wash station and carefully remove the print from the build plate
- Replace build plate on Anycubic remembering to tighten hand screw
- Open transparent cover of cure station and place washed print in center
- Add desired cure time. 10 - 15 minutes is typical
- Start curing print
- Once your part is cured, it is ready to use