Bambu X1 Carbon Standard Operating Procedure

(last edited by Kenneth Ho on 7/18/25)

Description

The Bambu Labs X1C is an FDM 3D printer (Fused Deposition Modeling) with a build volume of 256 mm x 256 mm x 256 mm (10.08” x 10.08” x 10.08”). It requires training to use as well as an FBS reservation. There are four different hotends with different nozzle sizes available: the 0.2 mm, 0.4 mm, 0.6 mm, and 0.8 mm. More on this here.

Safety

High Temperatures! - The print head and build plate are heated to high temperatures during a print. Be mindful of this and try not to touch anything inside the printer while it is in a print.

The printer stays hot immediately after the print is finished and while it is paused. Please be careful when removing a print, and **DO NOT** attempt to clean the build plate before giving it adequate time to cool down. It is recommended to wait until the build tray reaches 60 C or below before you can begin working with it again.

Moving Parts! - When a print is ongoing, the print head will be moving at relatively fast speeds across the entire build plate. Be mindful not to put your hands or any other body parts into the chamber to avoid any sort of injuries.

PPE Requirements

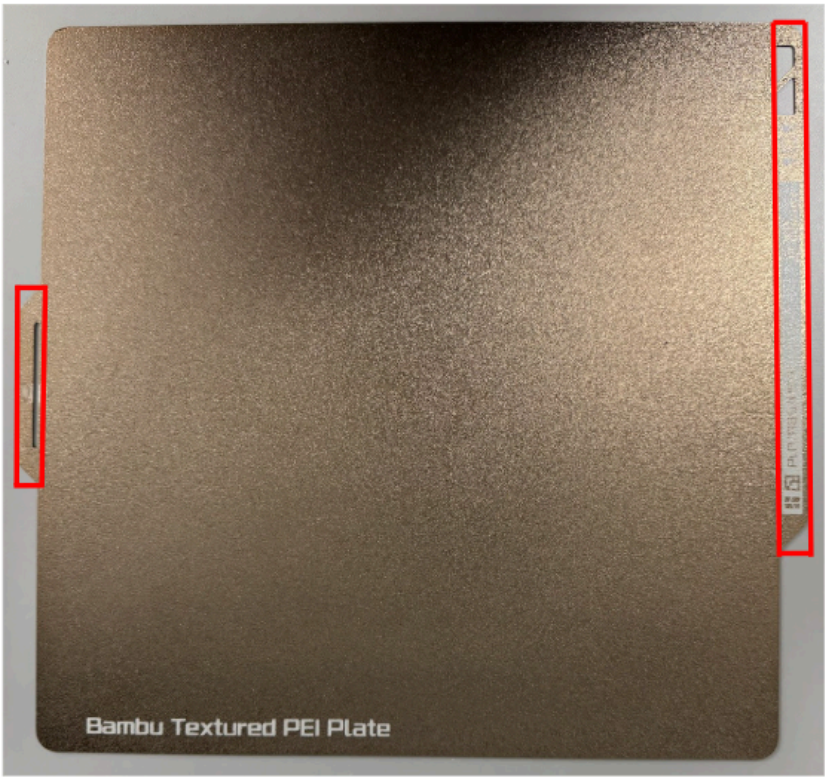
- Long pants
- Close-toed shoes

Pre-printing checks

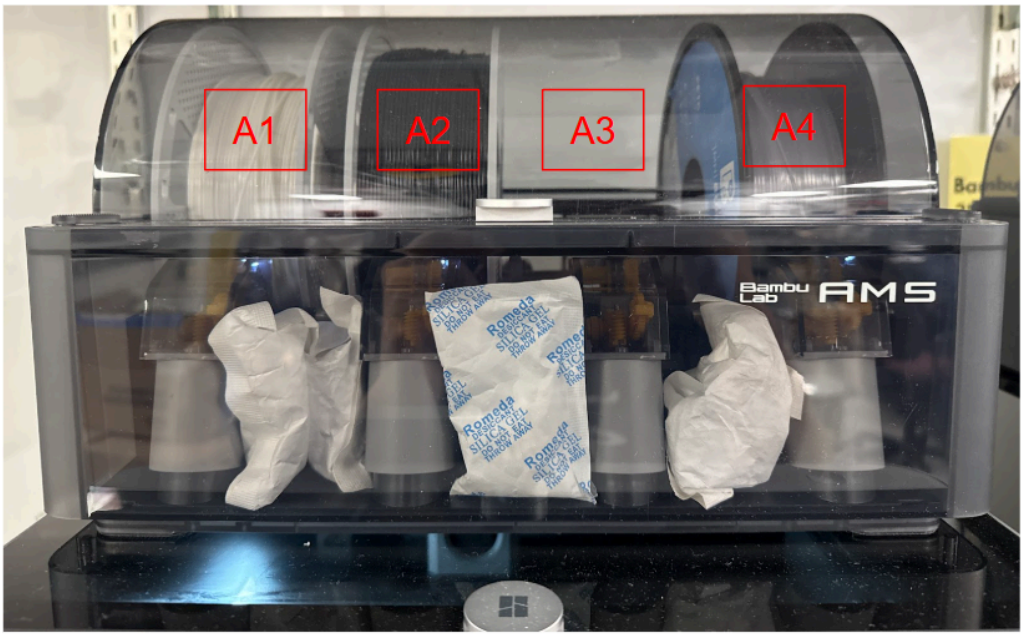
1. Before you come into the lab to start a print, it is recommended to set up a reservation on FBS to ensure you can use one of the X1C's during your allotted time. This will also allow you to check if the printer is available for use before you come in.

2. Remove any excess filament leftover from previous prints (if any). These would normally include things like a brim or the calibration lines that print on the edges of the build plate.

3. It is highly recommended to take the build plate out of the printer and bring it over to the sink to wash it with dish soap and water using only the soft side of a sponge. You can dry it with paper towels.
    a. Doing this can help with any adhesion problems your print may have.
    b. As a rule of thumb, **DO NOT** touch the base of the plate with your bare hands. Hold the build plate by the tabs on its sides to ensure no body oils get on the build plate.
    c. When putting the build plate back into the printer, make sure you line it up with the magnetic bed and place it with the smaller tab going into the printer and the larger tab facing out of the printer.



4. Check the material in the AMS to ensure the one you want to print with is available and in stock.
    a. Make sure there is enough filament in the spool for your print to complete. If the spool is running low on filament, replace it with a new spool.
        i. If we are running out of a filament in our storage, please contact the lab staff or the lab manager to restock.
    b. **If you need to switch out any filament in the AMS**, refer to step 2 in this page. Please change the filament type on the touchscreen after you change the filament in the AMS.

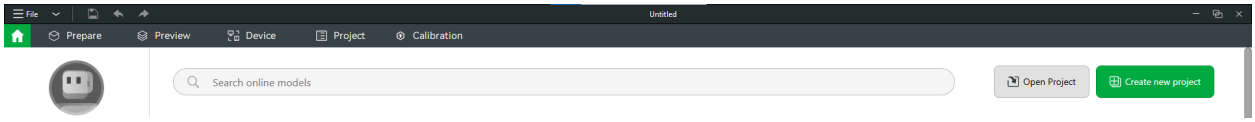


Setting up a print with Bambu Studio

3D printers from Bambu Labs use its own software called Bambu Studio. Through that software you can change any parameter of your print and directly send print requests to the printer. For more information about printing with these printers, you can visit the Bambu Labs Forum.

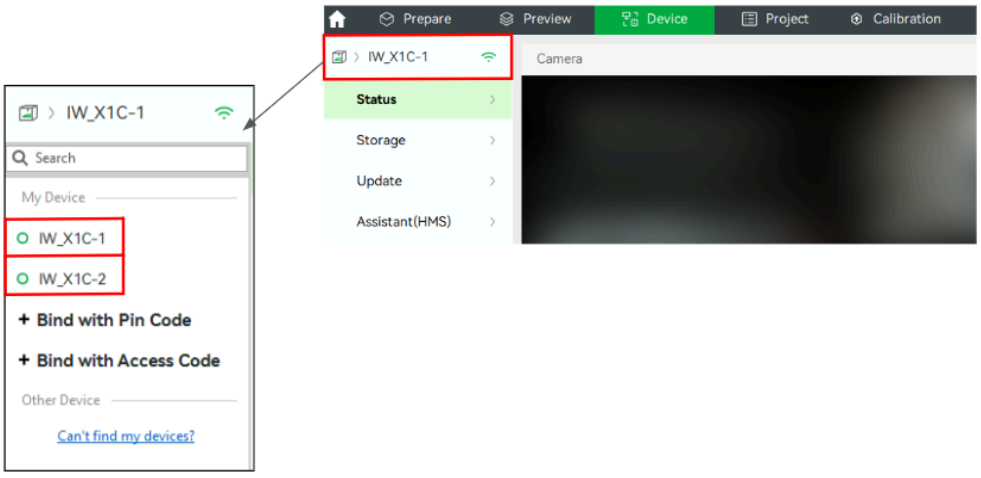
1) Open Bambu Studio.

2) Go to the homepage on the top left of the screen and select “Create New Project” on the top right of the screen.



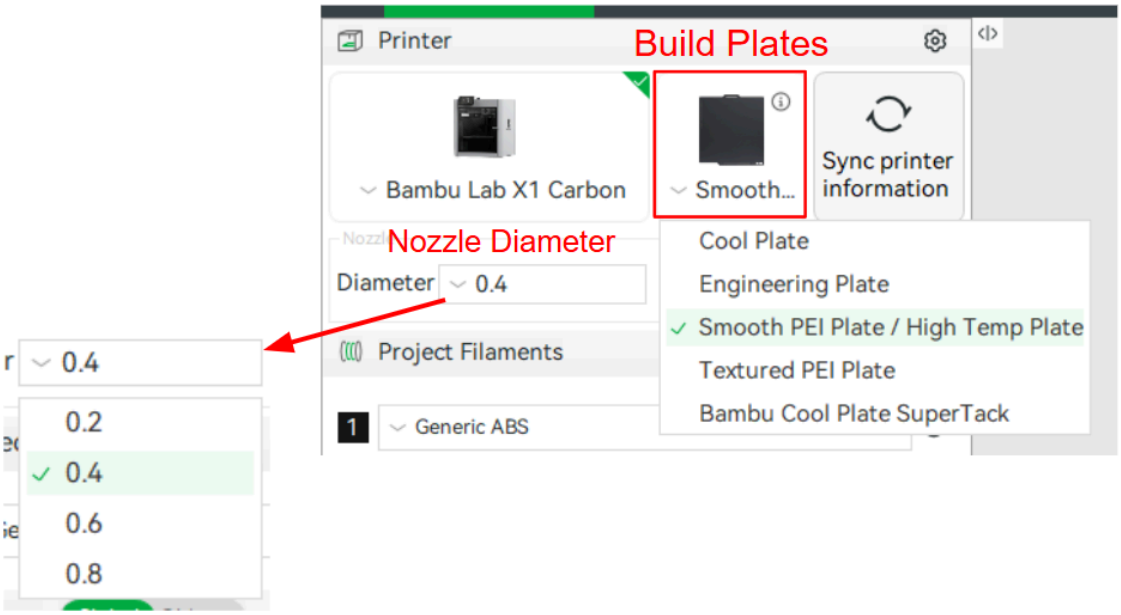
3) Import a file into the software. You can do so by either dragging the file into the software, or by going to File >> Import. You will need an STL file to set up a print. This software is also compatible with many other file types like STP, OBJ, AMF, and SVG. File types like .3mf and .gcode are also compatible. These file types are usually for your prints with fully adjusted print settings.

4) Select your printer, and set up the correct settings. Go to the “Devices” page and look in the top left corner to choose the correct printer you reserved on FBS (IW_X1C - 1 or IW_X1C - 2).



5) Go to the “Prepare” tab to begin preparing your print.

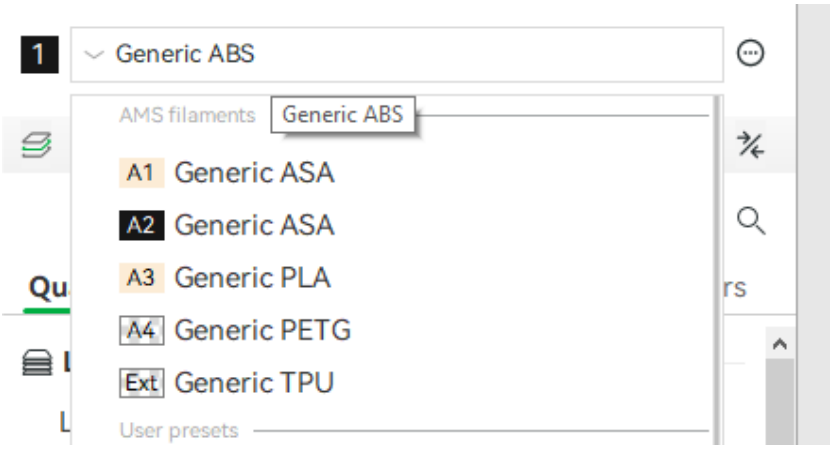
6) Choose the correct build plate. As of right now, we have 2 smooth PEI and 2 textured PEI plates.

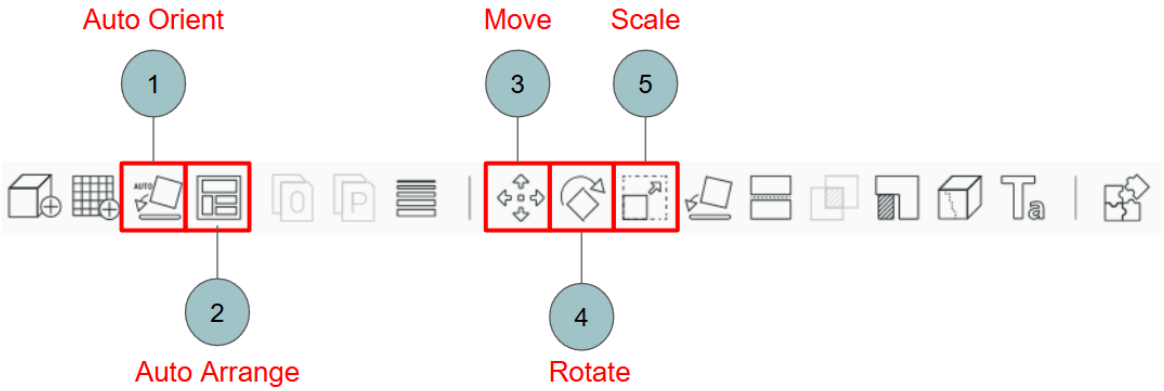


7) Select the **correct nozzle diameter** that is currently in use by the printer. Contact the lab manager or the lab staff to help switch this out if needed, or refer to this Bambu Lab WIki Page. More information about the nozzle and nozzle sizes in this section.

8) Choose your desired layer height. The options in the dropdown menu for the layer heights are premade recipes by the company that work well with the printer. When the 0.4 nozzle is in the printer (which is the default nozzle size), the 0.20 mm layer height is the standard setting.

9) Choose your material and AMS slot. The correct material should be displayed and match the AMS slot number that your material is loaded into the Bambu X1C. If not, the printer may not be connected correctly or you are using the wrong printer (refer to step 4)





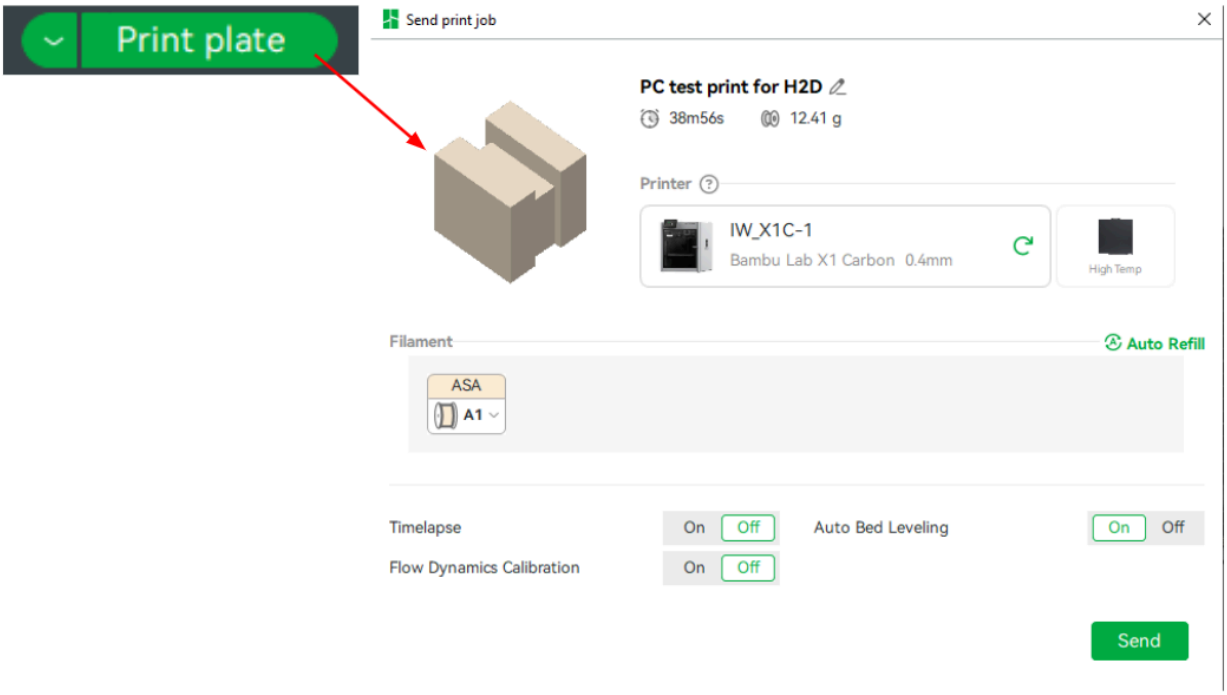
10) Set up the scaling, orientation, arrangement and supports as needed.
    a) The following refers to the figure above:
        (1) **Auto-orient:** This function will give you what the software thinks is the best orientation for your print to complete.
            (a) It will usually give you the best setup for supports, however you still want to think about specific features and printer limitations before you move on with your print.
            (b) For simple parts, this is usually the best option. If a print fails or does not generate the supports correctly, manual adjustment instructions can be found below.
        (2) Auto Arrange: This function will arrange your model near the center of the build plate, which is the hottest part of the printer, allowing for better adhesion and overall print quality.
        (3) Move: To move your part around in the software, click and drag it around. You can also select this “Move” function to use coordinates to move the part. The build plate on the software matches the build plate in the printer, so place parts accordingly.
        (4) Rotate: Rotate your part to get it to any orientation/configuration that you would like. Rotation is possible in all 3 dimensions, and you can do so by either dragging around the arrows or using the coordinates.
        (5) Scale: Scale your part by using the arrows or inputting the dimensions that you desire.
            (a) Bambu Studio uses mm for the dimensions in their software which can cause some inaccuracies or incorrect file imports if your model was created in inches/cm. You should always have specific dimensions in mind when you are setting up your print.
            (b) You can also scale the model by percentages if you wanted your model to be twice the size (200%) or half the size (50%) for example.

11) The settings under the “**Quality**”, “**Strength**”, and “**Speed**” tabs can remain unchanged as the default settings should suffice for your print.

12) Under the “**Support**” tab, click enable support to auto-generate supports for your print. Supports are useful if you have any overhangs as it prints sacrificial material that will be used to print those overhangs on top of the supports. There are two types of supports that you can select; tree supports and normal supports.
    a) Tree supports connect with less parts of your print, creating a tree branch pattern for the supports. Typically, these are easier to remove than the normal supports and are recommended to use for your prints.
    b) Normal supports tend to make contact with a large area of your print that has the overhang. This can sometimes lead to more difficult post processing, but it can also be the support type you need to use for certain overhangs and features.

13) Under the “**Others**” tab, you can add a brim. This is recommended if you have a larger print that has a lot of area on the build plate to help with adhesion. You should also use it if you are just generally having adhesion problems. To add a brim, go to the drop down menu on “Brim type” and select “Outer brim only”. After that is done, you can adjust the brim width. The default width of the brim is 5 mm, however larger brim widths give stronger adhesion.

14) Once your settings are adjusted to your liking, select “Slice plate” on the top right corner to get a sliced view of your print. With this sliced plate, you can see how the individual layers will be printed, but you can also see what the print would look like with supports and/or a brim. In this plate, you can also see the print time and the amount of filament that will be used. Use the scroll on the right and bottom of the screen to view the inside of your print and verify settings

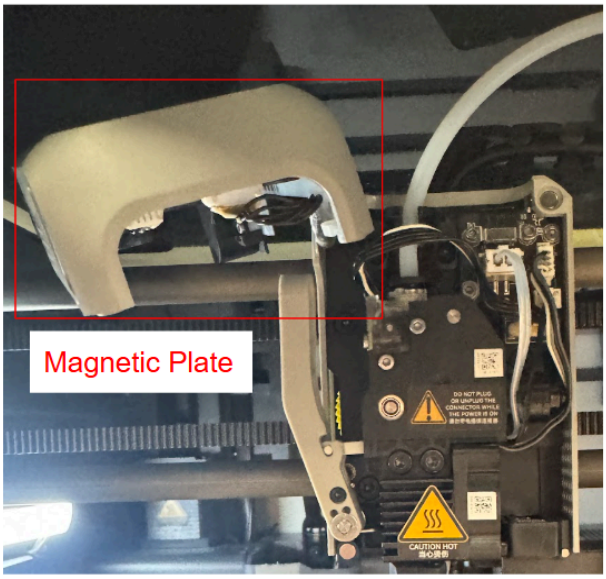


15) Press “Print plate” (can only be done after slicing) and ensure that auto bed-leveling is on. Double check that “timelapse” and “flow dynamics calibration” are turned off. On this screen, you can also double check to see if you have selected the correct printer, filament, and build plate.

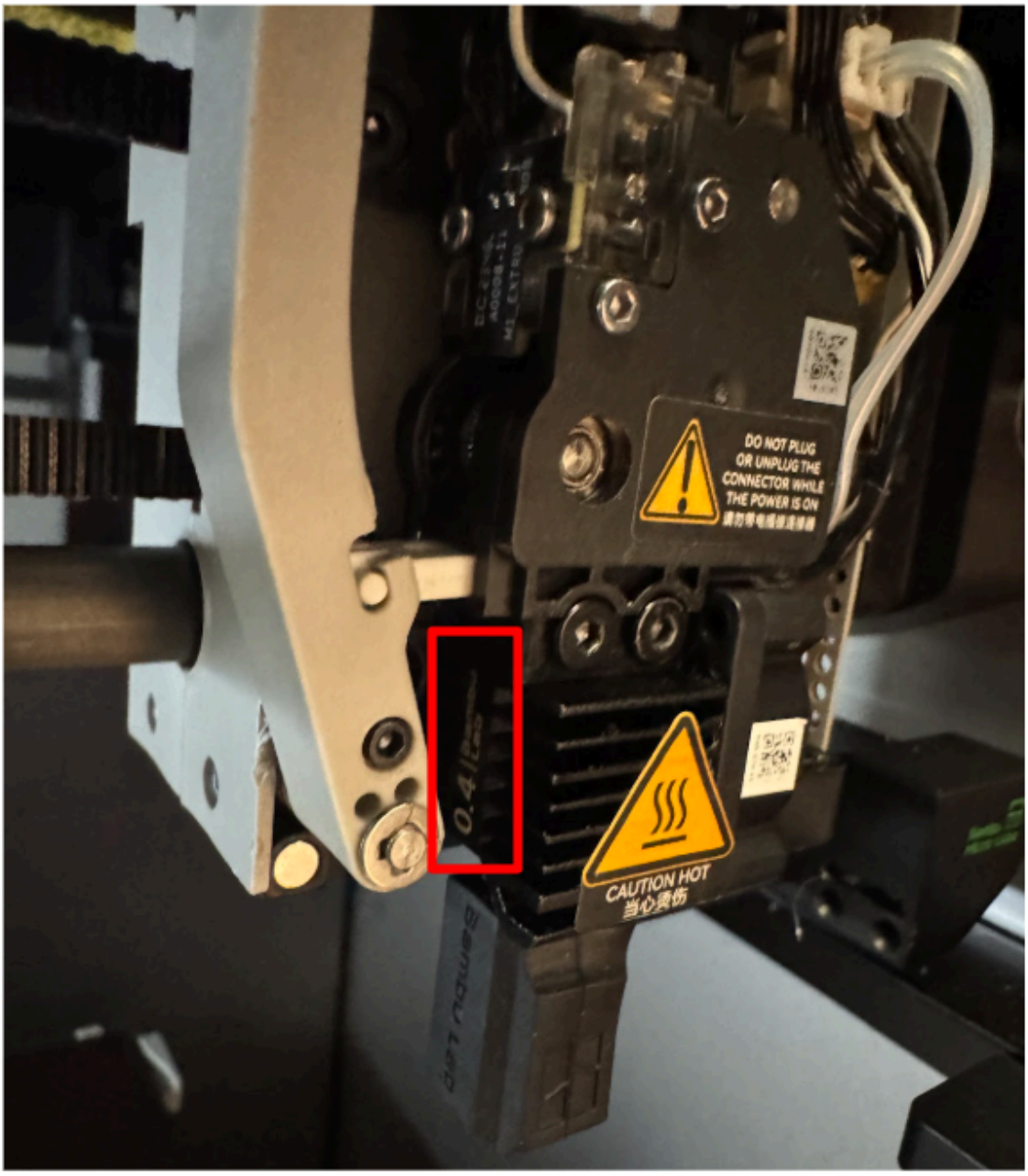
Hotend nozzle

1) There are 4 different nozzle sizes that can be used with the printers. Each nozzle setting allows you to select different layer heights for your prints. The nozzles that we have in the lab are the 0.2, 0.4, 0.6, and 0.8 mm nozzles. Normally, the 0.4 mm nozzle will be in the printers, however we have labels for the nozzles on the sides of the printer.
    a) The 0.2 nozzle allows you to use the smallest layer height, where the 0.8 nozzle allows you to use the largest layer height.
    b) In general, smaller layer heights give you higher quality features with the tradeoff of a longer print time. The opposite is true for larger layer heights, it may not print your smaller features with as high of quality, but it makes for a shorter print time.
    c) Again, if you want to switch out the nozzles, refer to this helpful wiki page.

2) In order to check which nozzle is being used in the printer, start by moving the print head closer to the center of the printer so that it’s easier to work with. Move aside the magnetic plate on the print head to access the inside. It will be connected by a wire so you cannot fully take it off, but you can move it slightly out of the way so that you can access the inside of the print head. As shown in the picture, you can **carefully** place the magnetic plate on the xy-gantry to keep it out of your way.



a) Move the print head towards the right of the printer so that you can access the left side. Looking at the left side of the extruder, you can see the exact nozzle type that is being used.



(NO INTERNET) Manual Printing with MicroSD card

In order to print **without internet**, or through direct connection with the Bambu X1C, please refer to the following instructions.

1) Locate the MicroSD card in the slot found in the bottom on the right side of the touchscreen. Press it in once to release the card.

2) Use an adaptor to connect the microSD to a lab computer, or personal device. Upload your file into the SD card under an appropriate name. You can find a microSD adapter in 2448 either connected to the waterjet or connected to the computer next to the waterjet.

3) Insert the microSD card back into the slot in the Bambu X1C touchscreen.

4) Locate the “Print Files” folder on the touchscreen. This will be on the screen when you select the Home button. Your file should be the first one on the list, however if you do not see it there scroll through the files to look for the file name that you assigned.

5) Once you have selected your file, double check that you are using the correct plate and nozzle. Once again, deselect “TimeLapse” and “Flow Calibration”, and make sure “Bed Leveling” is selected. Check the filament in the AMS and select the right slot.