Tab 1

Bambu H2D Carbon Standard Operating Procedure

(last edited by Colin on 10/6/25)

Description

The Bambu Labs H2D is an FDM 3D printer (Fused Deposition Modeling) with a swappable dual nozzle tool head. It has a build volume of 325 mm x 320 mm x 325mm (12.80” x 12.60” x 12.80”) with a single nozzle and a build volume of 300 mm x 320 mm x 325mm (11.81” x 12.60” x 12.80”) with dual nozzles (See more info about printable range). It requires training to use as well as an FBS reservation. The Innovation Workshop’s H2D uses 0.4 mm hotends. 0.2 mm, 0.6 mm, and 0.8 mm hotends will be available in the future. More on hotends can be found in the hotend section.

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

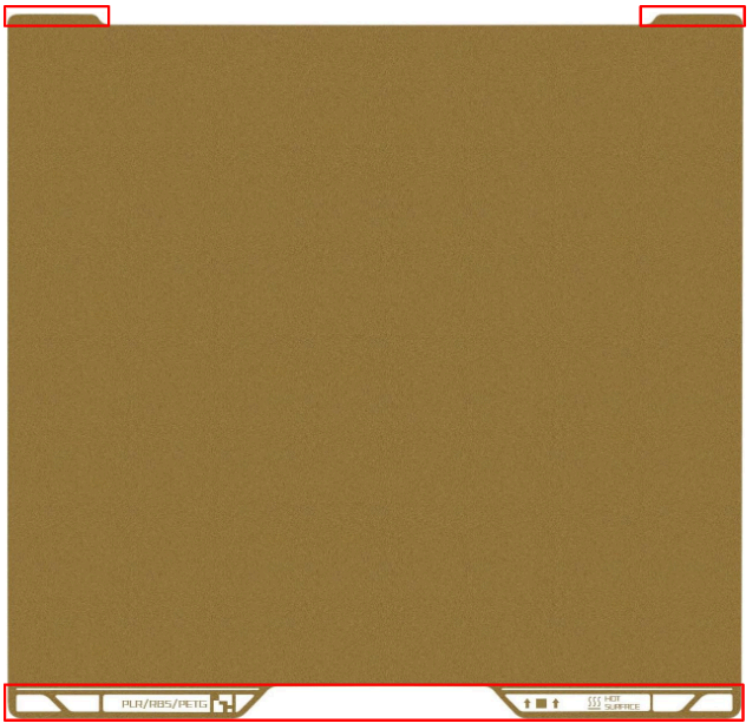
Pre-printing checks

Before starting a print on the H2D, please review the following:

1. It is recommended to set up a reservation on FBS, prior to setting up a print in IW. This ensures you can use one of the H2D's during your allotted time. It will also allow you to verify the printer's availability.

2. Remove any excess filament leftover from previous prints (if any). These would include brims or purge lines left on the build plate (see below image). This can be done by using the 3d printed scraping tools. Avoid touching the build plate with your hands to maintain print adhesion.

3. It is highly recommended to clean the build plate for better adhesion. This can be done as follows:
    a. Lift the buildplate by the tabs (See image below, tabs boxed in red) to remove from the magnetic heat bed.
    b. Bring the build plate to the sink and wash with dish soap and water. Scrub the buildplate with the soft side of the sponge.
    c. Dry with paper towels
    d. As mentioned earlier, **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 (See image below, tabs boxed in red).
    e. Place the build plate back into the printer, make sure you line it up with the magnetic bed and place it with the text side facing the door/you.



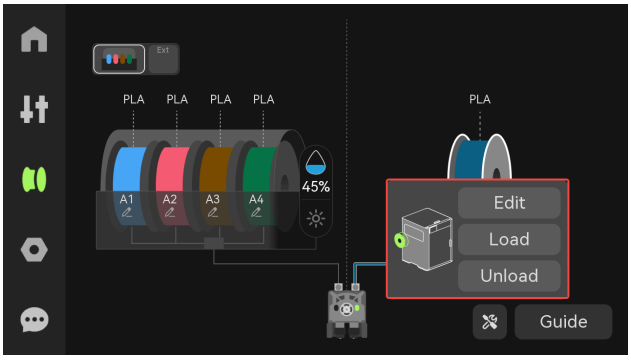
4. Check the material in the AMS to ensure the one you want to print with is available and in stock.
    a. The AMS (Automatic Material System) is located on top of the H2D. It holds 4 filament spools for swapping materials and multi-material printing. Refer to the Bambu Lab's Introduction to AMS for more info.

b. 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.
c. If we are running out of a filament in our storage. This is located under the X1C's please contact the lab staff or the lab manager to restock:

d. If you need to switch out any filament in the AMS, refer to the AMS setup guide (start from step 2). Please change the filament type on the touchscreen after you change the filament in the AMS.

5. A Creality filament dryer (located to left of H2D) holds a spool used for dual printing (left nozzle)

    a. To swap out the left nozzle filament, navigate to the  icon within the H2D display.
    b. Select the spool labeled as “Ext” and select “Unload”. This unloads the current spool in the extruder



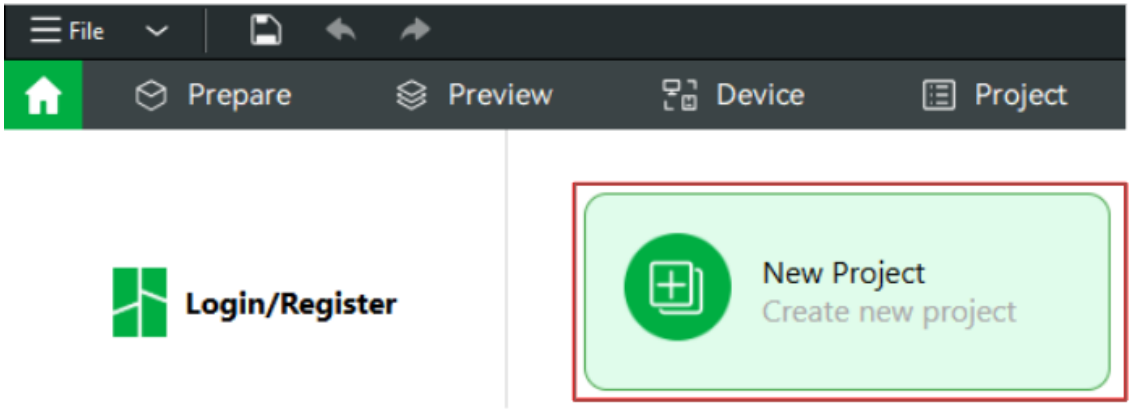
    c. Remove the spool from the filament dryer
    d. Place desired spool into the filament dryer, and feed the filament up the bowden tube until it can’t be pushed further. (See below images)

Print Setup with Bambu Studio

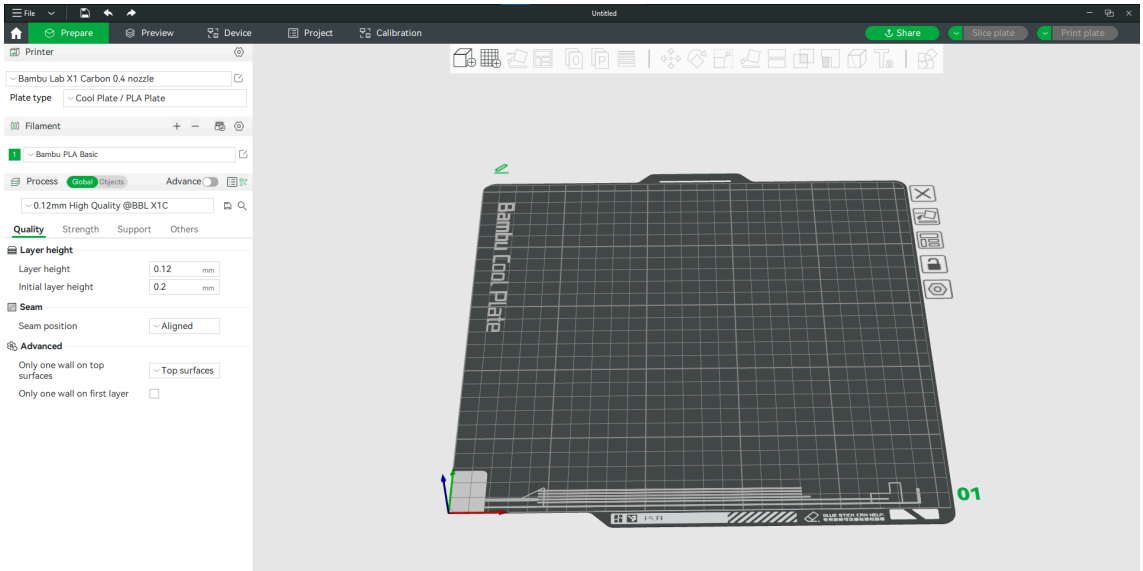
Bambu Labs 3D printers use Bambu Studio as its slicer. This is a software that allows users to change print parameters and send prints to the printer. General discussion/troubleshooting can be found on the Bambu Labs Forum. Here is a general guide for Bambu Studio:

1) Open Bambu Studio. This is Bambu Studio icon:

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

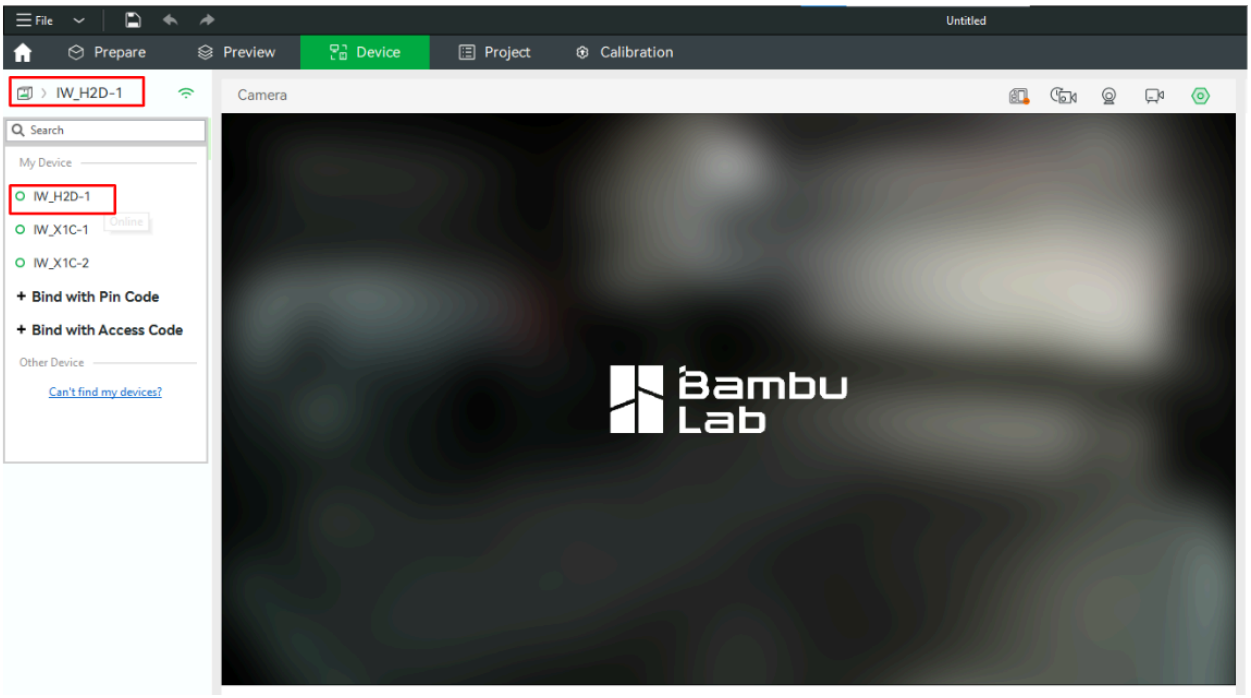


3) Bambu Studio will now display the print area along with print settings. See Bambu Studio keyboard shortcuts/camera controls



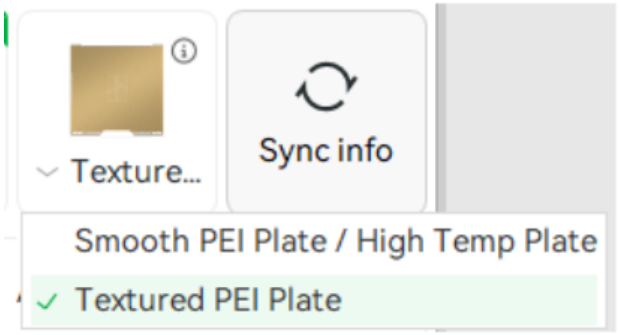
4) Import your 3D model by dragging the file into the Bambu Studio window, or by going to File >> Import. This software is compatible with file types such as .stl, .stp, .otp, .amf, and .svg. File types like .3mf and .gcode are also compatible, and are usually for prints with fully adjusted settings.

5) Navigate to the top left of the “Devices” page and select the printer you reserved on FBS (IW_H2D).



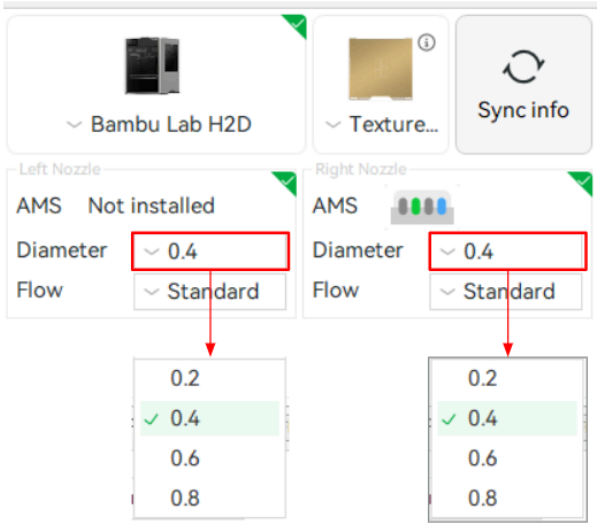
6) Go to the “Prepare” tab to begin preparing your print.

7) Choose the correct build plate. We have 1 smooth PEI and 1 textured PEI plate. See these links [more info on build plates]



Build Plate

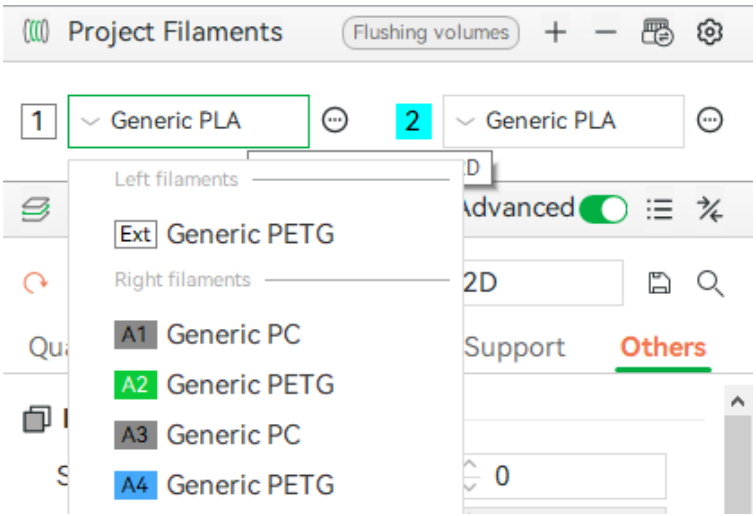
8) Select the **correct nozzle diameter** used by the printer. The H2D currently only uses 0.4 mm nozzles. Contact the lab manager or the lab staff to help switch this out if needed, or refer to hotend replacement guide. More information about the nozzle and nozzle sizes in this section.

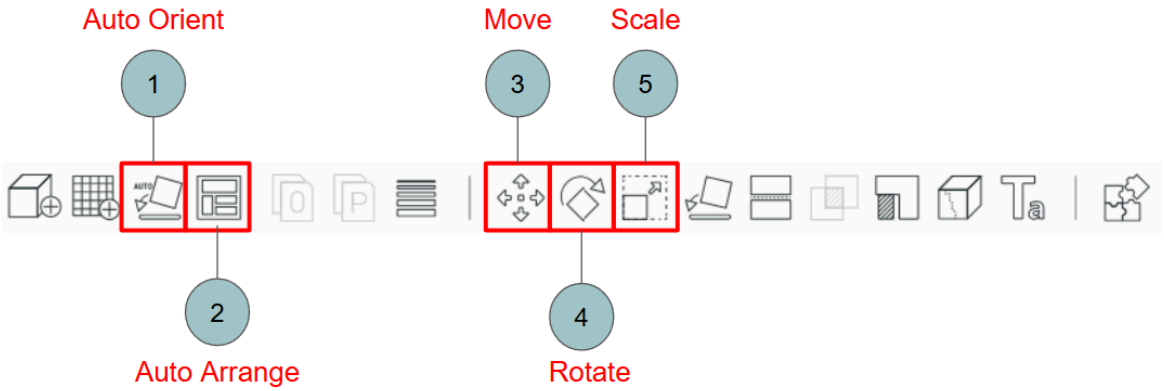


Nozzle Diameter (mm)

9) Choose your desired layer height. The options in the dropdown menu for layer heights are presets by Bambu Labs that work reliably. The 0.2 mm layer height is standard with a 0.4 mm nozzle. Refer to the Bambu Studio layer height guide for more information.

10) Choose your material and AMS slot. The displayed materials should be in the AMS with the associated slot number. If materials are not shown, the printer may not be connected correctly or the wrong device is selected. (refer to step 5). Materials with a prefix “A” are stored in the AMS. Those labeled “Ext” are in the filament dryer. Click the “+” icon to add additional printing material.





11) 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.

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

13) 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. See the supports guide for more info.
    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.

14) 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.

15) 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

16) For multi-material printing, please refer to the following guides:
    a) Bambu Labs Guide
    b) Youtube Guide

17) 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.

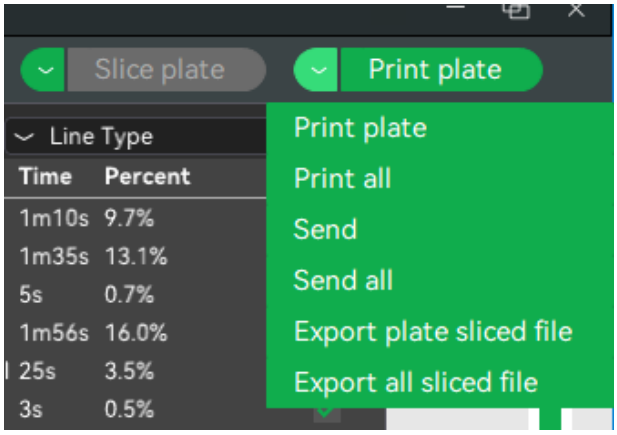
Hotend

1) The Bambu Lab printers use 0.4 mm size nozzles. Additional 0.6 mm and 0.8 mm nozzles will be available in the future.
    a) 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.
    b) Again, if you want to switch out the nozzles, refer to this helpful wiki page.
    c) The nozzle size can be viewed from the front of the toolhead:

(NO INTERNET) Manual Printing with USB flash drive

In order to print **without internet**, or through direct connection with the Bambu H2D, please refer to the following instructions.

1) Insert a USB-A flash drive into the computer/your device
2) After slicing, navigate to the arrow to the left of “Print plate” and select “Export plate sliced file” or “Export all sliced file”. This allows you to send a .gcode to your flashdrive



3) Disconnect your flash drive

4) Locate the USB-A slot found in the bottom on the top left side of the touchscreen. Insert your flash drive here.

5) 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.

6) 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.