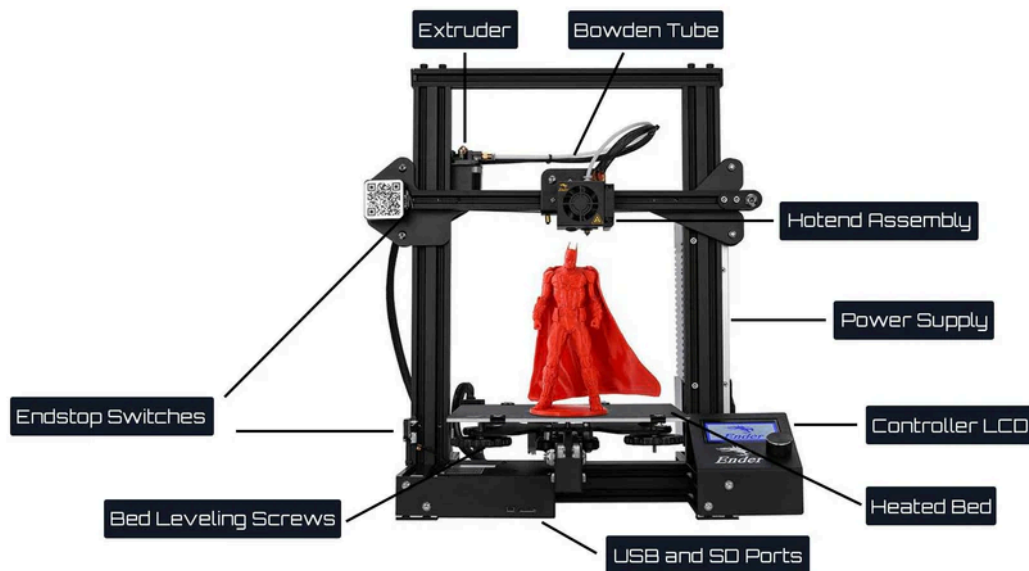




FDM Printers

Basic Overview

FDM printers heat and print with hot plastic. They produce a lower amount of detail than other printing methods (SLA and SLS) but are ideal for fast production of items in which high resolution is not critical, like prototypes, figurines, non-moving parts, and cosplay pieces. They print with a variety of plastics, but most often are used with PLA which is highly durable. These printers require .GCODE files which must be saved onto microSD cards



Extruder: mechanism for pushing filament to hot end/nozzle.

Bowden Tube: the tube the filament feeds through to the hot end.

Hot End: Composed of nozzle and hot-end fan. Melted filament is deposited from the nozzle.

Power Supply: The power supply on the side of the printer

Controller LCD: The screen used to control the printer

Heated Bed: The hot bed, which the print sticks to and builds upon

USB Ports: Where the USB and micro SD's are loaded into for file transfer

Bed Leveling Screws: Screws beneath the bed used to level or stabilize the bed

Endstop Switches: Switches placed on each axis to make sure the printer stays within its limitations and doesn't break its own hardware.

Safety Precautions

Plates and Nozzle get very hot! While the printer is in operation, preheating, or cooling, don't touch the bed or the hot end.

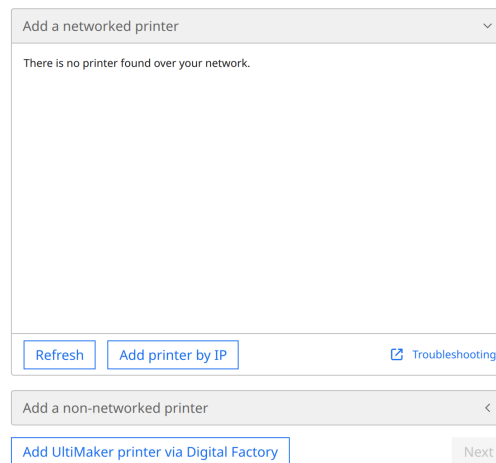


Step by Step

Creality Products: (CR10 and Ender3)

1. Start by creating an account for [Ultimaker Cura](#), the slicing software used to prepare a 3D model for printing. The app is available to download from Ultimaker's website.
2. Once the account is activated, open the Ultimaker Cura app. Follow the dialogue options to add a Non-Ultimaker and Non-Networked printer.

[Add printer](#)

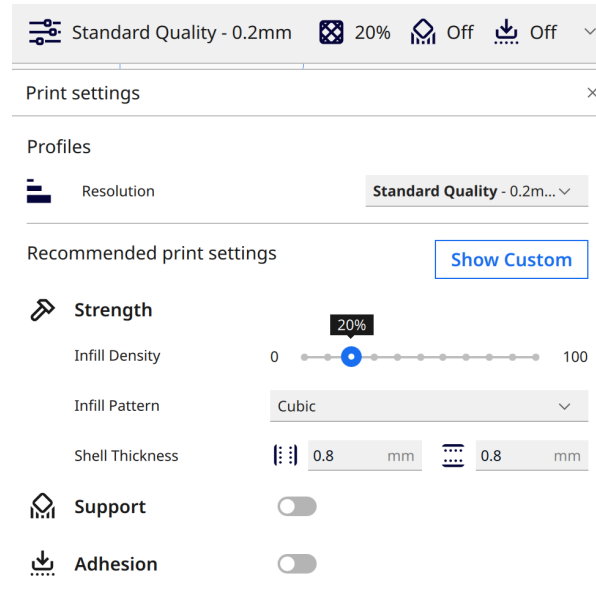


3. From the drop down list, navigate to Creality and then the appropriate model of printer.
4. Open a file. Ultimaker Cura can import STL, OBJ, X3D, 3MF, BMP, GIF, JPG, PNG files, but most often users will be working with STL or OBJ.
5. Use the navigation tools. Once the file is imported, a control panel will appear on the left edge of the screen. The tools you'll use the most are the move, scale, and rotate. The mirror tool reflects the object across one of the axes, the model settings tool brings up a mini panel of the settings, and the support blocker turns off supports. You'll almost never need the last three.
 - a. Move Tool: Move the object along the plate by clicking the arrows and dragging it. You can also input the distance of the translation by axis. Make sure the object has contact with the build plate.
 - b. Scale Tool: Scale the object by clicking on the file and dragging the scaling arrows or by inputting the desired dimensions or scaling percentage. Check the uniform scaling option in order to make the scaling consistent in X, Y, and Z axes. Ensure the part is inside the build volume.



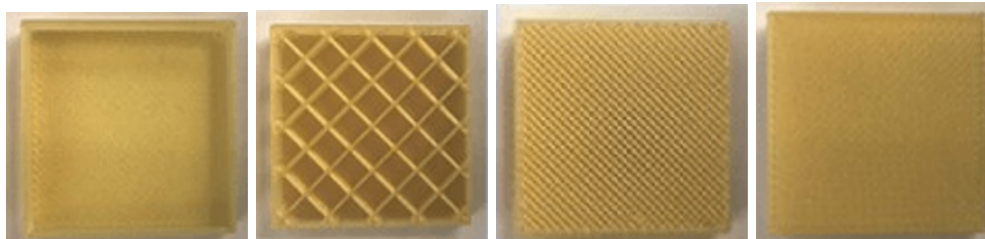
Idea Realization Lab - User Reference Sheet

- c. **Rotate Tool:** Rotate the file by holding and dragging on the colored lines around the file. The angle will be changed by 15 degree increments. Ensure that the part is in contact with the build surface.
6. **Print settings.** Once your item is placed, open the settings panel by clicking the small arrow in the upper right corner of the screen.



- a. **Infill:** The infill is the internal material supporting the object. The infill percentage can be adjusted from 0% (empty) to 100% (solid). Higher infill results in a *stronger* and *heavier* part, with a *longer* print time. Lower infill results in a *lighter* part with a *faster* print time. You'll usually never need more or less than 20%.

0% 20% 50% 100%

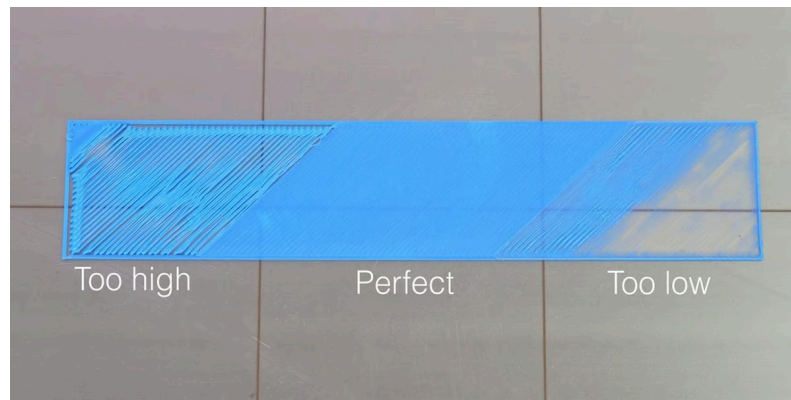


- b. **Supports:** Supports must be printed anywhere the object is not supported either by contact with the build plate or with the lower part of the print itself. Each layer of FDM printing requires something to rest on, so failure to include supports will result in a failed print. Supports are removed after print is finished.
 - c. **Build Plate Adhesion:** Build plate adhesion should always be turned on. It forms a skirt around the object that makes it easier to adhere to the plate, and can be removed once the print is finished.



Idea Realization Lab - User Reference Sheet

7. Slicing and exporting the file. Once print settings are established, simply click Slice in the lower right corner. Slicing separates the object into the layers that the printer will produce, and exports a new file type: .GCODE. Save this file to a microSD card.
8. Print. Insert the microSD card into the printer. Make sure you're using the same model printer that you set in Ultimaker Cura, or the object may not fit in the printer build volume.
 - a. Use the control wheel to navigate to Print from Card. Select by clicking the wheel/button once.
 - b. Navigate again to your file and click to print.
 - c. While the print bed and nozzle come to temperature, you may wish to apply a light layer or hairspray to the print bed for extra adhesion.
 - d. Observe the first few layers of your printer to ensure it's off to a good start. While it may still fail in later stages, an unlevel bed - the most common cause of failed prints - will be visible early on. This is what your first layer should look like. If it does not, the print bed is not level.



9. After the print is complete, remove the print from the print bed by gently prying with a putty knife. Remove and discard the supports

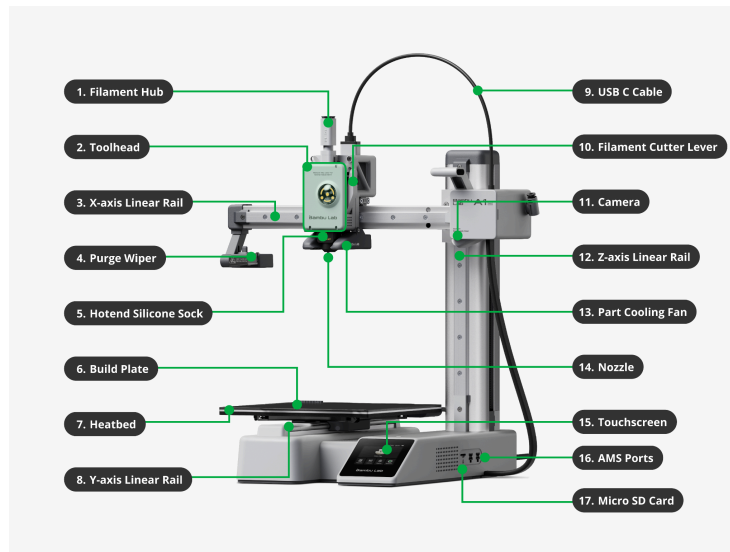
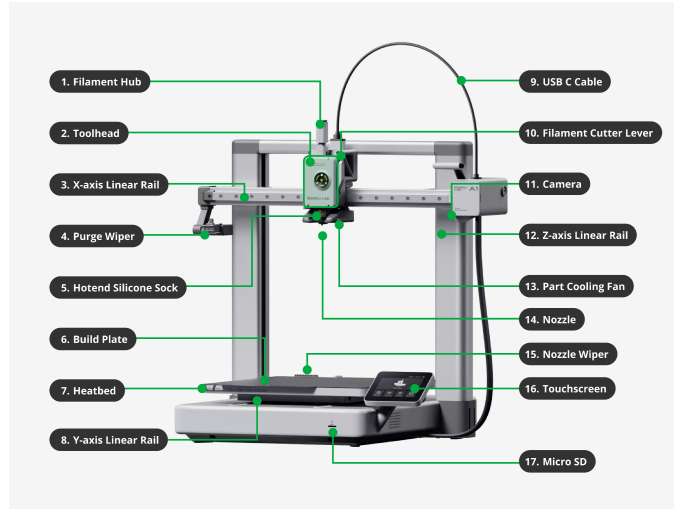
Bambu A1/A1 Mini/X1C

Basic Overview

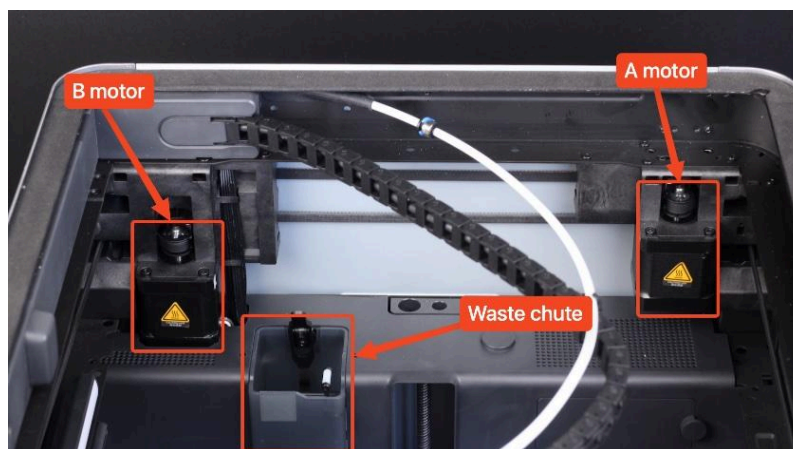
The Bambu printers, unlike the Crealitys, require less routine upkeep, as they are able to self-level and WiFi connect. They generally print faster than the Crealitys, and can use the same PLA filament. The prints can be sent via WiFi or with a MicroSD, and the sliced file is a .3mf. We use Bambu Studios to slice it.



Idea Realization Lab - User Reference Sheet



The Bambu Lab X1 uses a CoreXY motion system controlled by two stepper motors. The X and Y stepper motors work together to move the printhead.

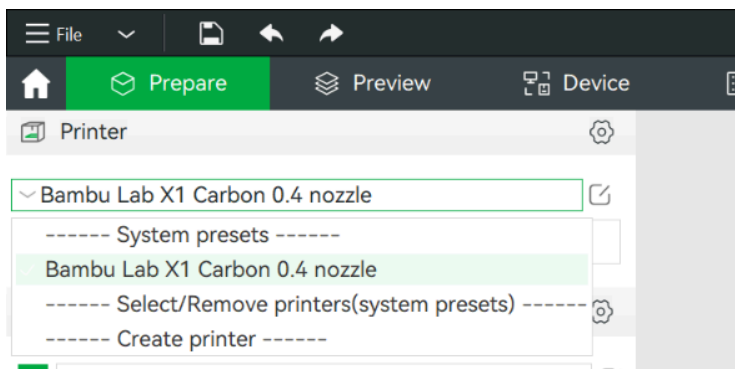




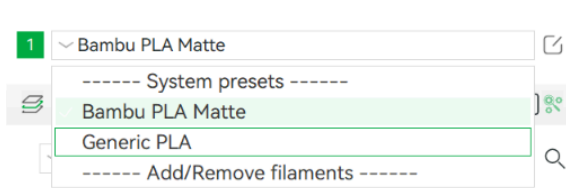
Every stepper motor has an independent belt that is connected to the print head, so a pair of belts is used to control its position. Having a CoreXY motion system allows the X1 to print much faster compared to a traditional cartesian printer because the weight is lower, which is important when it comes to printing fast.

Step by Step

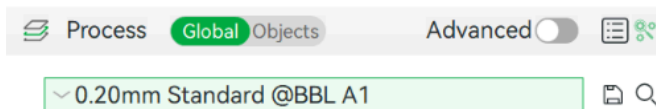
1. Download [Bambu Studio](#) for your personal device, and make an account.
2. Hit "create new project" and select the type of printer in the drop down menu by clicking "select/remove printers (system presets.) Then, select the 0.4 mm nozzle of either the Bambu Lab A1, or Bambu Lab A1 mini.



3. Select plate type: Textured PEI plate
4. Select filament type ("Generic PLA").



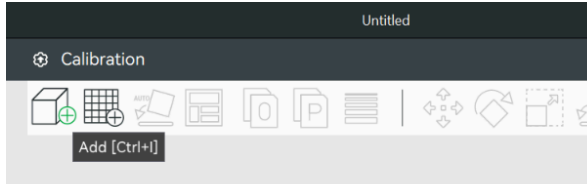
5. Under Process, choose the layer thickness from the dropdown menu. The standard is 0.20mm Standard @ BBL A1.



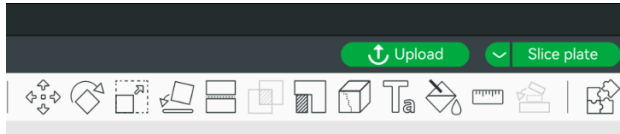
6. Upload your .stl file by clicking the "add" button.



Idea Realization Lab - User Reference Sheet

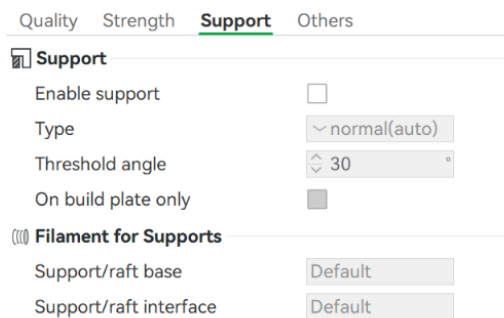


7. Make sure that it is positioned correctly, using the options on the top of the screen.



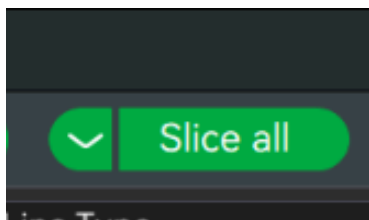
We have, in order, the move tool, rotate tool, and scale tool. Those will be the tools you use the most.

8. Go to the supports menu



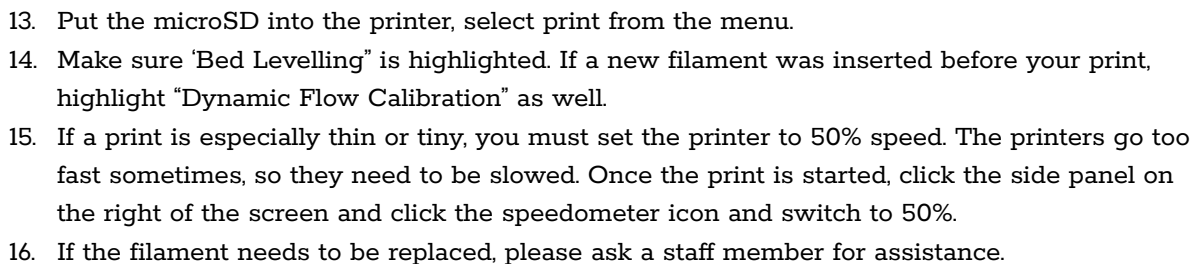
Hit "enable support" and let it generate. Go to the preview tab to see where the supports generate.

9. Go to the Slice option and enable "slice all"



Then, slice the file.

10. A menu will appear. This will tell you the total time, filament and grams. Please keep these in mind when determining how much filament is left in the machine. It may need to be replaced to accommodate the size of your print.

[illegible]

What do I do if filament for my 3D print runs out?

1. What should I use to make sure my print doesn't fall off of the printing bed?
This may sound funny but use hair spray! It works great with making sure your print stays in place! Only use hairspray on the creality products though. The bambu doesn't need it!
2. The nozzle is moving in the right direction but no filament is coming out!
The 3D printing filament is either jammed, or the filament was not pushed far enough into the nozzle. Stop the print and find an IRL staff member to unclog the nozzle.
3. The filament isn't adhering to the build plate. What should I do?
It's most likely a leveling or temperature issue. Find a staff member to level the plate or change the heat of the bed.