

3D Printing

Basic Use and Safety



250 First Street | Woodland, CA 95695
www.cityofwoodland.org/squareone | 530.661.5989

Square One Etiquette:

In an emergency, dial 9-911

Library General and Behavior Policies apply in Square One.

The most up-to-date policies are always available for review at the front desk.

- Be **Safe**: Pay attention, follow posted procedures, and ensure proper materials are being used including eye/ear protection in the woodshop.
- Be **Curious**: Experiment, try new things, watch what other people are doing.
- Be **Respectful**: Keep your chaos contained, clean up after yourself, don't touch other people's work without asking, don't interrupt.
- Be **Aware**: Alert staff if equipment isn't functioning properly, follow staff instructions, pay attention to your project.
- Be **Prepared**: Bring your photo ID and library card, wear closed-toe shoes, secure loose or dangly objects (i.e. hair, jewelry, keys), start cleaning up 30 minutes before closing time.

Notes:

3D Printer Policies:

Square One is *not* a professional 3D printing service or 'print farm'. Please be respectful of our machines, 3D printing materials, employees and other patrons, so that everybody has a chance to use our equipment.

You may bring your own filament, provided its diameter and material type are approved by staff. [Generally, PLA filament is well tolerated by our printers; PETG requires special care and handling to avoid clogging the equipment.] However, we cannot be held responsible for spools of filament left on machines past closing hours.

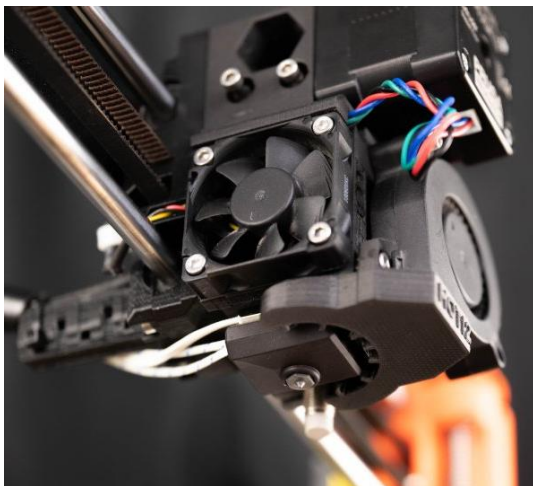
Only one print at a time is allowed. Do not begin multiple prints at the same time on different printers. Once your print is complete you may begin another print if no one else is waiting to use the printer.

Before starting a new print, patrons must fill out a Maker Magic slip, which includes the patron's name, library card number and filename of object printed.

If your print will run past closing time you may pick up your completed print at a later date. You may not remain in Square One after closing to wait for your print to be completed. Make sure to notify Square One staff that you will be leaving your print to run after closing and provide your contact information.

Completed prints will be held for pickup behind the counter for one week. Square One is not responsible for the quality of prints left unattended. Square One is not responsible for prints not picked up within one week.

3D Printer Safety Rules:



- Very high temperatures are used to melt the filament - never touch the metal parts of the extruder under any circumstances.
- The extruder often moves quickly from one position to another. Long hair, jewelry, or loose clothing may become caught on the printer arm as it moves. This could result in personal injury, a ruined print or damage to the machine. Please ensure there is nothing on your person or in the nearby area that might become caught on the machine before you start your print.
- Never reach into the machine after beginning the printing process without first pausing your print.

The 3D Printing Process:

3D printing is an additive process. The printing process builds up an object layer by layer using a fine thread of plastic filament which is laid down in layers following a path that is dictated by the computer. The computer program moves the arm with the extruder into position and the extruder lays down the filament in that location.

Software:

In order to be printed on a 3D printer an STL file (.stl) needs to be processed by a piece of software called a "slicer," which converts the model into a series of thin layers and produces a G-code file containing instructions tailored to a specific type of 3D printer. Each 3D printer will use its own particular slicing software, though the Taz printers use CuraLE, while the Prusa uses PrusaSlicer, and the Bambu X-1 Carbon uses Bambu Studio.

Scaffolding and Structural Concerns:

Because a 3D printed object is created by adding layers of plastic on top of existing layers there are some limits to the physical shape of the objects that can be created using 3D printing. The printer can only print on top of an area it has already printed- it can't print a part of your design that will be suspended in space. (Imagine a tree with broadly spreading branches - that's not a great design for 3D printing.)

Most 3D printers can handle designs that incorporate angles up to about 45 degrees. A good rule of thumb is that if your design goes *out* more than it goes *up*, chances are it won't result in a successful print without using scaffolding to support the design or employing some other sort of structural support during the printing process.



(image credit: medium.com)

Many existing designs have this support built into the design but if you are working with your own design or customizing an existing design, you will need to take this into consideration.

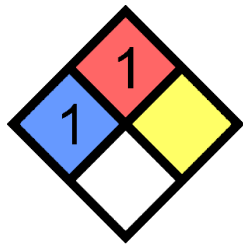
How much will it cost?:

Square One currently does not charge for 3D printing materials; however, they are *not* free, so a donation, while not mandatory, would be greatly appreciated. Therefore, please be mindful of the amount of 3D printing material you use in your prints. The slicing software will tell you how much material an object will require, once the object is finished slicing. We recommend that you keep the amount of material used for each individual print under 200 grams whenever possible.

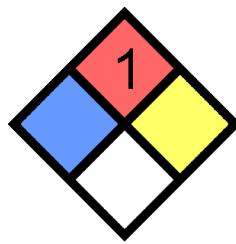
A rule of thumb: a typical, brand-new \$25 spool of filament has about 1000 grams of material on it (+/- 10%). So, a print requiring 200 grams would therefore require 1/5 of a full spool of filament to print.

If you wish to print something that requires more than 200 grams, we strongly advise that you bring your own filament for the job. You can purchase suitable filament from most online outlets. Some recommended brands of PLA that work well with our printers: Elegoo, Polymaker, eSun, Hatchbox, Matterhackers. Please be sure to have staff approve the material you wish to use.

Material Safety and Approved Materials:



PLA



PETG

Due to material out-gassing and hot-end clogging concerns, we currently only allow PLA filament to be printed on any of our printers without supervision. PETG has a very high tendency to clog, and can only be done well on printers with enclosures; this particularly means the Mini as it has a full enclosure and an extruder designed to facilitate repair of clogs. *Please* ask a Square One employee to assist with printing with PETG filament. (Please do not print PETG with our Bambu Labs XI Carbon, despite it having an enclosure, as the print mechanism may not be easily serviceable.)

Approved materials: PLA, PETG (*only* with supervision). These materials have a relatively low VOC level while printing.

Unapproved materials: ABS, PC, PEEK, ASA, HIPS, ...

These materials have very high VOC (volatile organic compound) and/or micro-particulate emission levels while printing, and may constitute a danger to health without appropriate filtering and ventilation.

Some General Recommendations for Settings:

These are general guidelines rather than requirements. Part of the “fun” of 3D printing lies in tweaking these settings in order to get the best print quality. If you’re working on a prototype you may end up making multiple adjustments to your settings in order to overcome printing problems and fine tune your print quality. These recommended settings are a good place to start. Most of these settings are already set to these recommendations by default; however, when starting a new print, it is a good idea to ask a staff member to review the settings before beginning a print.

Layer height recommendation:

1=.0992, 2=.1984

Shell thickness: Usually, you use a multiple that is 10x the nozzle diameter of the printer you are using. The Lulzbots all use 0.5mm nozzles, so you will want to use a multiple of 5 with them. Meanwhile, the Prusa uses a 0.4mm nozzle, so a multiple of 4 is appropriate. The Ultimaker uses a 0.25mm nozzle, so a multiple of 2.5 is necessary.

Enable retraction: Usually you want to do this to prevent ‘stringing’.



“The purpose of retraction is to relieve pressure from the melt zone so that filament isn’t being forced through the nozzle during non-print moves. The correct amount is the minimum amount required to reduce the most stringing on your part. Some machines and hot ends require more retraction than others, and each material has different requirements. In general, though, it’s unlikely you should need more than 5mm or less than 1mm”

(<https://www.matterhackers.com/articles/retraction-just-say-no-to-oozing>).

Enabling retraction will activate this feature at the default setting for the printer you are using. If you experience ‘stringing’ in your print this setting can be further adjusted manually.

Top/bottom thickness: should be $n \times$ layer height

Temperature:

PLA=205-210

PETG=230-240

Support type:

Everywhere: Usually, if support is needed, it is best to have the printer add support everywhere, unless you have a specific reason not to add it.

Platform adhesion type:

Brim or raft (usually raft – it makes a better base but increases the print time and amount of filament used slightly)

Diameter 1.75 (2.85mm for the Ultimaker 2+ and Lulzbot Mini)

Flow rate- 100.00



Ultimaker 2+ ("Oscar")

Our Ultimaker 2+ is a long-serving, perfectionistic but user-friendly stalwart that uses 2.85mm filament and has a very tiny diameter (0.2mm) nozzle. While it is our slowest 3D printer by far, it makes great quality prints with a little patience and experimentation.

(Image credit: ultimaker.com)

Loading filament:

Turn the dial on the front of the printer until **MATERIAL** is highlighted on the display. Push the dial button to select the material menu.

Turn the dial on the front of the printer until **CHANGE** is highlighted on the display. Push the dial button to select change.

The display will read **Heating print nozzle** and the printer will begin to remove the old filament from the feed.

Straighten the end of the filament to be loaded and trim end to give a clean edge.

Load the filament spool onto the back of the printer, making sure the filament comes off the roll in a counter-clockwise direction.

The display will have the word **Ready** highlighted. Press the dial button to continue.

Select the material you are loading by spinning the dial. Press the dial button to continue.

The display will read **Insert new material** once the nozzle is preheated.

Feed the filament gently into the feeder until the gears grab the filament and the filament is visible in the tube. (You may need to cut the tip of the filament at a 45-degree angle to help the gears grab it.) Select **Ready** and press the dial button to continue. The filament will begin to feed more quickly.

Once the filament begins to be extruded from the nozzle select **Ready**. Remove extruded filament with tweezers.

Printing with the Ultimaker 2+:



Please bear in mind that the very small nozzle diameter is a double-edged sword: due to its very small diameter, it takes much longer to print than the other printers. Therefore, it is practical to use this printer to print relatively small objects with moderate-low infill levels (~25%) only, otherwise you will very easily run past the allotted 24 hours per print job.

On the desktop computer launch  Ultimaker Cura (so-called 'Blue Cura').

Select the printer (ultimaker2) from the machine menu.

Plug USB stick containing your .stl file into the desktop computer.

Click "load model".

Select the USB storage location, find your .stl file and click "open".

At this point you can make changes to any settings in Cura.



Do not use Lightning infill. Lightning infill has a very high tendency to clog the extruder, due to it using extremely short retractions!

Remove the SD card from the Ultimaker2+. Inset the SD card into the USB SD card adapter. Plug the SD card adapter in to the desktop computer (or the USB hub attached to the computer).

To save G-code: Once a model is sliced, click 'Save to Removable Drive'. Give your file a file name and save as type *.gcode. Click to save.

Remove the SD card from the SD card adapter and put it back into the Ultimaker 2+.

Use the dial to select the print option. (If the print option is not visible, select return to locate the option.) Scroll through the print menu using the dial until you see your file name. Hit the button to select your file.

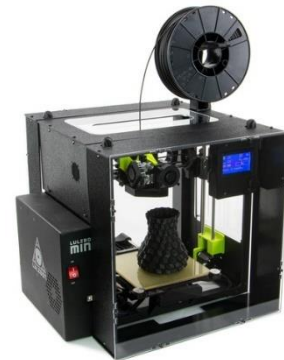


Lulzbot Mini, Taz 6 (“Cookie Monster”) and Workhorse (“Purple Tentacle”)

The true workhorses of our fleet, our heavily-upgraded Taz printers feature large print beds (with 6/Cookie’s print bed being a flexible, magnetic sheet that is removable for easy part removal), 0.5mm nozzles, custom firmware with BLTouch automatic bed leveling, and moderate print time and detail quality. They both also use 1.75mm filament. (Image credit: IT-Works 3D)

Despite their superficial similarities, they are different enough that trying to print something sliced for one Taz on the other, will result in a failed print.

The Mini is an older, small-bed printer that has seen several upgrades to “keep up” with the rest of the machines. It uses 2.85mm filament, and has an enclosure to help it print high-temperature filament such as PETG. (Image credit: lulzbot.com)



Loading filament:

To change the filament, you can use the **Change Filament** option on the printer itself. Using the control knob, highlight **Change Filament** and press the center of the knob to select it.

Then, select the kind of filament you wish to load; however, if you are replacing a high temperature filament with something cooler like PLA, then select the filament that is already on there.

Follow the prompts on the printer. It will briefly recalibrate, and then heat up the extruder. When it beeps at you (Cookie’s a little soft-spoken, so you might have to listen closely), unclip the idler and remove the filament that is currently loaded.

Make sure new filament has a nice clean edge and insert it into the feed hole in the extruder body. Then reclip the idler. Press the control knob on the printer when you’ve loaded the filament. Remove any excess filament from the hot end with tweezers.

Printing with the Lulzbots:

On the desktop computer launch  Lulzbot Cura (so-called 'Green Cura').

Select Lulzbot Mini, Taz 6 or Workhorse from the machine menu. **Make sure you take note of the printer selected.** If you try printing using the wrong printer, your print will fail!

Load a model into Cura from your USB stick, or from the L: drive. (Don't forget to remove your USB stick when you're done loading files.)

Manipulate the print and settings as needed within Cura. It's a great idea to note of the settings you've changed within Cura as well, so you don't accidentally sabotage the next patron.



Do not use Lightning infill. Lightning infill has a very high tendency to clog the extruder, due to it using extremely short retractions!

When you are ready to slice, hit Slice print. When it is finished slicing, remove the memory card from the desired printer, insert the card into the card reader, wait a moment or two, and click **Save to Removable Drive**. (Take note of the name of the file.)

When it is saved, you may hit the Eject Removable Drive button, wait a moment for it to say you can remove it, remove the SD card from the adapter and re-place the SD card in the printer.

When you are ready to print, select **Print From Media** on the printer, and choose the file you wish to print.

It will ask if you want to start the print. If you've selected the correct file, hit Print; otherwise, hit Cancel and try it again. (Sometimes the control wheel moves more than one space with each rotation, so it is quite possible to select the wrong file.)



Prusa i3 MK3S

This user-friendly, 1.75mm filament printer is capable of very good quality prints thanks to its 0.4mm nozzle, and has a removable magnetic bed that facilitates easy removal of prints. An aftermarket Slice Mosquito hot-end has made it a much more reliable machine than ever before.

[Image credit: Prusa]

Loading filament:

To change the filament, you first must select **Preheat Filament** on the printer. Using the control knob, highlight **Preheat Filament**, and press the center of the knob to select it.

Then, select the kind of filament you wish to load; however, if you are replacing a high temperature filament with something cooler like PLA, then select the filament that is already on there.



Due to its lack of an appropriate enclosure, the Prusa isn't especially great with high-temperature filament, so we recommend sticking with PLA for this printer.

When it gets up to temperature, you can select **Unload Filament**. Follow the prompts on the printer. It will briefly recalibrate, and then heat up the extruder. When it beeps at you [the Prusa has a very loud, annoying beep!], unclip the idler and remove the filament that is currently loaded.

Make sure new filament has a nice clean edge and insert it into the feed hole in the extruder body. It will automatically grab the filament and begin loading it.

It will shortly ask if the filament is extruding with the expected color. Select Yes if it is; otherwise, hit No. When it's done extruding, remove any excess filament from the hot end with tweezers.

Printing with the Prusa:

On the desktop computer launch  **PrusaSlicer**.

On the menu bar at the top of the window, click **Import > Import STL/3MF/STEP/OBJ/AMF...**

Select the file you wish to load into PrusaSlicer, which may be located on your USB stick, or within the L: drive. (Don't forget to remove your USB stick when you're done loading files.)

Manipulate the print and settings as needed. It's a great idea to note of the settings you've changed within PrusaSlicer as well, so you don't accidentally sabotage the next patron.



Do not use Lightning infill. Lightning infill has a very high tendency to clog the extruder, due to it using extremely short retractions!

When you are ready to slice, hit Slice Now. When it is finished slicing, remove the memory card from the desired printer, insert the card into the card reader, wait a moment or two, and click **Export G-Code**. (Take note of the name of the file.) Select the location of the SD card, and hit Save.

When it is saved, you may hit the Eject button on the window that pops up, wait a moment for it to say you can remove it, remove the SD card from the adapter.

When you are ready to print, insert the SD card in the printer. It will automatically begin reading the card. (If it does not bring up a file list, select **Print from SD**.) Choose the file you wish to print, and select it by pressing the center of the knob.



BambuLabs X-1 Carbon

Our newest exciting addition is a very advanced, high-speed printer capable of multi-color, multi-material printing with great detail. It is a quite different experience from our other machines, however, so some of the steps to use it are strikingly different. One obvious difference from the other printers, is that it uses a touchscreen to control the device. Additionally, due to its enclosure it is a great choice for using high-temperature filaments. (Please ask staff to assist with PETG.)

(Image credit: 3D Herndon)

Loading filament:


If you need to change the filament, tap the Settings icon, and then touch **Unload**. The AMS will wind the filament back up onto the spool. Carefully pull on the filament while also re-spooling it on the filament spool. You may use the small holes on the sides of the spool to keep the filament in place. Lift away the old spool, and insert the new spool in the now open slot. Carefully take the end of the filament, and cut the tip at a 45-degree angle before inserting it into the hole where the filament goes in.

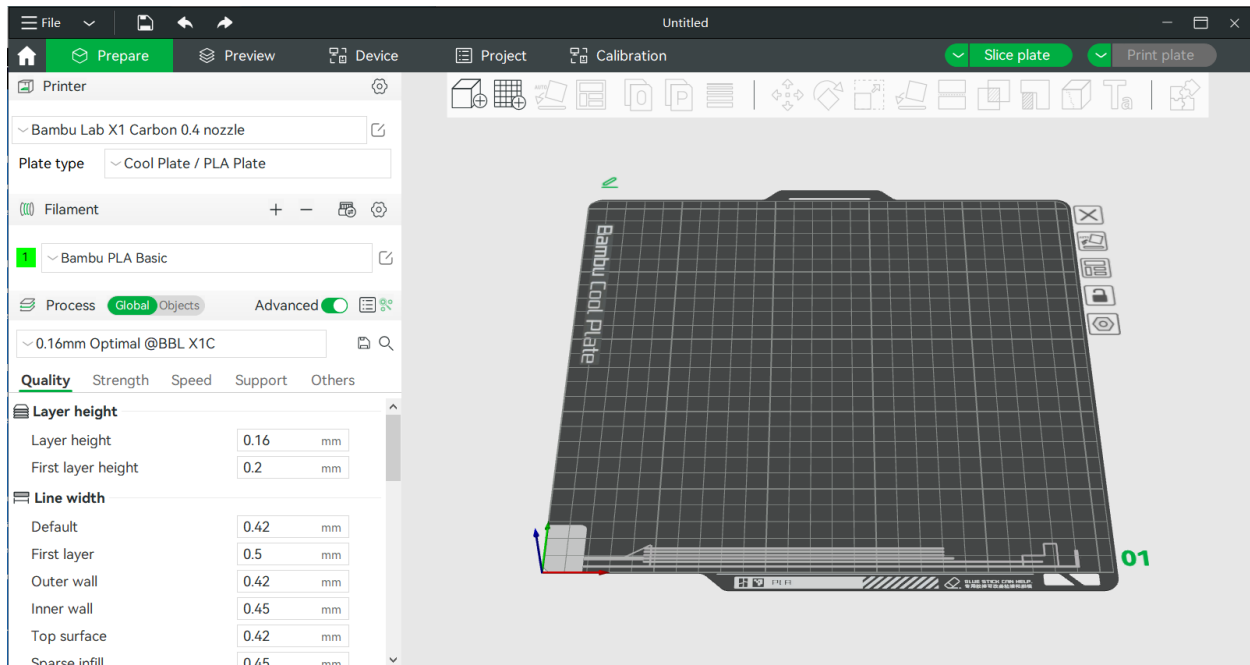
When you insert the filament, the machine should automatically ‘take’ the filament and begin loading it into the extruder. If a BambuLabs branded spool is loaded, it should also automatically determine the filament type and color; if it doesn’t, you can tap the number of the loaded slot and manually change the color and filament type from there.



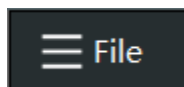
It is important to note that the AMS box is *not* designed to work well with filament wound on cardboard spools. We strongly recommend you use filament wound on plastic spools to avoid jamming.

Printing with the X-1 Carbon:

On the desktop computer, open up  **Bambu Studio**.



When it is loaded, dismiss any windows that may pop up additionally, and click the Prepare tab in the upper left corner of the program's main window.



Load a file by using the File hamburger menu icon in the top left, select Import 3MF/STL/STEP/SVG/OBJ/AMF...



Do NOT use 'Load Project', this will try to load a project file and NOT your model. The difference between your model and a project file, is your model is not a project file according to the program.

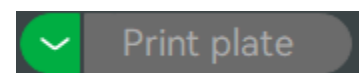
From there, you may select the file you wish to load, and hit Open.

Manipulate the print and settings as needed. It's a great idea to note of the settings you've changed within BambuStudio as well, so you don't accidentally sabotage the next patron. Note that the placement of most settings is quite different compared to Cura and PrusaSlicer, you may have to go menu-hunting to find the settings you wish to edit.

When you are ready to slice, hit Slice Plate in the upper right-hand corner of the window.



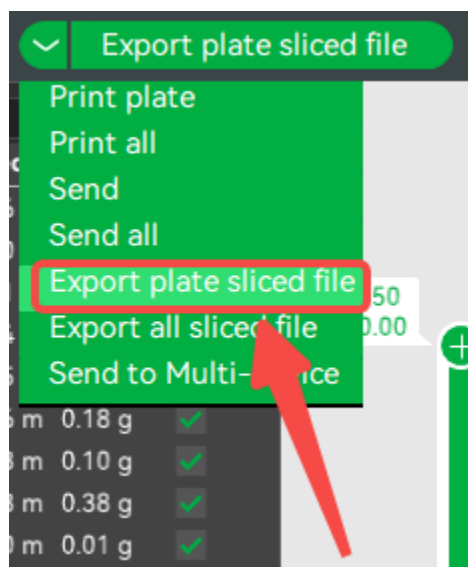
When it is finished slicing, remove the memory card from the desired printer, insert the card into the card reader, wait a moment or two, and press the down arrow right next to the Print Plate button.





But please *don't* hit 'Print Plate' – our computer does not have any connectivity to the printer beyond the SD card!

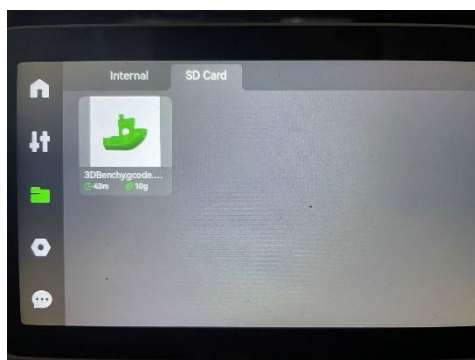
You want to change this to **Export plate-sliced file**, so select **Export plate-sliced file** from the drop-down menu that appears. Click **Export plate-sliced file**, select the location of the Micro SD card, and hit Save. (Take note of the name of the file.)



When it is saved, you may hit the Eject button on the window that pops up, wait a moment for it to say you can remove it, remove the Micro SD card from the adapter.

When you are ready to print, insert the Micro SD card in the printer.

Gently tap the screen of the 3D printer to wake it up. Tap the Folder icon (third icon down from the top, at the left of the touchscreen).



After a few moments, a graphical list of various sliced objects will appear. (Usually, the newest files will be listed first.) Tap the object you wish to print.

(You may have to tap the 'SD Card' folder icon toward the top center, if you are not getting a list of objects.)

Depending on the filament(s) loaded in the AMS box, another menu may appear. From here you can assign the colors/filaments to use for each selected color of the print. Tap the filament(s) you wish to change, and choose the appropriate slot as indicated inside the AMS box. The number of the slots increases from the left, starting at 1 on the far left, and ending with 4 on the far right.

When you're ready to start, tap the **Print Now** button. When printing with PLA, the machine will notify you that you might need to leave the door open while printing; we don't actually recommend leaving the front door open as it can easily catch on other patrons walking past the equipment.

(Image credits: <https://wiki.bambulab.com/en/software/bambu-studio/studio-quick-start>)