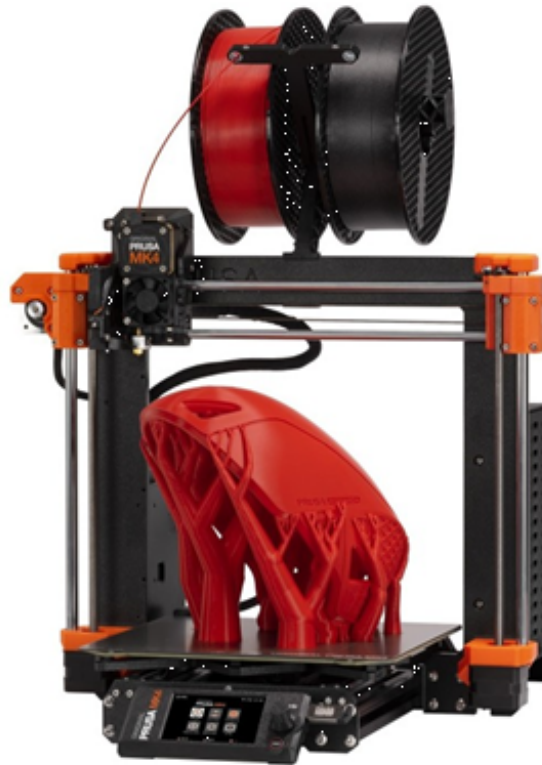


GETTING STARTED ON THE 3D PRINTERS



Bambu Lab
AMS



Get to Know 3D Printing

What is 3D printing?

- Our 3D printers use 'Fused Deposition Modeling' or FDM for short. It means the printers extrude a thermoplastic filament in a series of layers over a build plate to create a three-dimensional object.
- Think of our 3D printers as a fancy hot glue gun.
- The filament melts and draws with the melted plastic filament.

Materials

- We offer two types of filament at Creativity Commons—PLA and PETG.
- PLA - most common, great for most models.
- PETG - higher melting point, food-safe, more durable.

➡ Structuring a three-dimensional object in its physical configuration from its digital form.



3D printing and additive manufacturing

3D CAD FILE
.STL file format

CAM - SLICIGN
.GCODE file format

3D Printing

FINAL - PHYSICAL
OBJECT



Process of 3D Printing

The Process

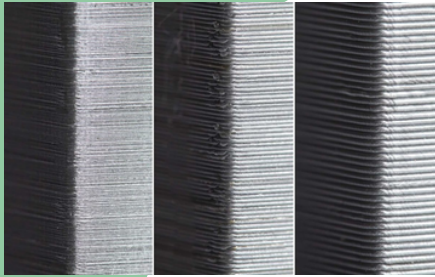
Design - Find a design on Thingiverse.com, Printables.com, Makerworld.com, or another site.

Slice - Slicing turns the design into a G-Code, or instructions, the 3D printer understands.

Print - Load material (filament) into the printer and the printer will follow the instructions sent via the computer.

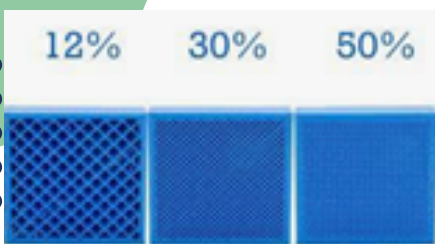


3D Terms to Know



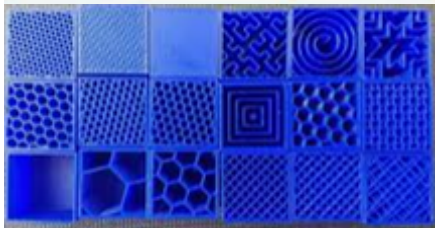
Layer Height

- Printed parts are made by stacking layers of plastic on top of one another.
- You can make the layer height smaller to create smoother looking prints. This option does take longer to print.
- You can make the layer height larger for less smooth prints that print faster.



Infill

- Most 3D printed parts are primarily hollow.
- The grid pattern that fills the inside of a design is called the infill.
- You can choose how much infill to use for stronger parts and change what patterns you want.



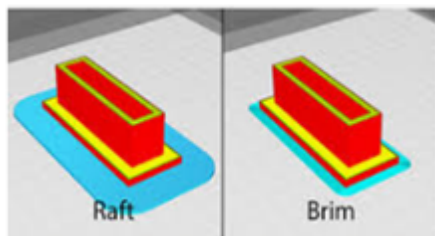
Support Material

- Support material is used to reinforce layers that would otherwise have nothing to hold them up.
- For instance the letter T, has no plastic underneath the arms, thus creating an overhang. An overhang is when there is no plastic beneath a portion of a design.
- For best results, add supports to your designs.
- Support materials easily break off after printing.



Brim

- Brims are additional loops of plastic on the outside of a part.
- Brims are used to help prevent large parts from peeling up from the build plate.
- After printing, brims can be trimmed off.



Raft

- Rafts are a few layers of plastic laid before the design prints. A raft allows for better adhesion for the whole print since the raft adheres to the print bed, and the print attaches to the raft.
- A raft goes all the way under the print and consists of multiple layers, whereas a brim is only one layer and is laid on the outside of the print.

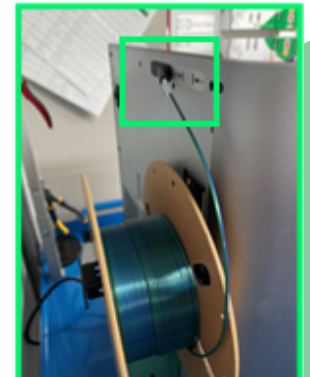
Bambu – Load Filament

To load or unload filament, touch the spool icon on the left menu on the screen. Tap the spool icon again and choose Load or Unload, depending on which you need to do.

The hotend will begin heating, and once it's up to temperature, will cut the filament. The screen will give you step by step instructions on what to do next.

When unloading, carefully pull the filament out of the machine while re-spooling it on the filament spool - you don't want it to tangle. Use clips to hold the filament in place.

When loading, hang the spool on the holder at the back of the machine and push into the tube until it reaches the extruder. Ensure that the machine begins to pull it in. Confirm that on the machine when it asks.



AMS Bambu - Load Filament

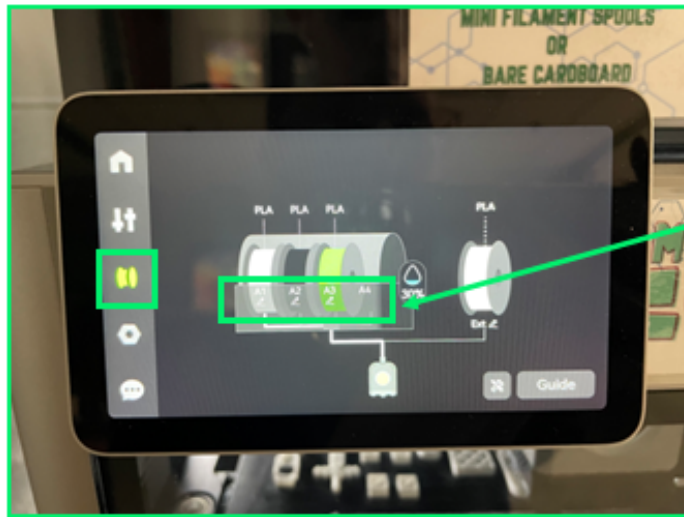


****IF PRINTING WITH MULTIPLE COLORS, THE FILE NEEDS TO BE SET UP TO DO SO. SOME FILES COME PREPARED THIS WAY. IF THEY'RE NOT, YOU'LL NEED TO CHOOSE THE COLOR PAINTING OPTION IN THE TOP TOOL BAR.****

AMS BAMBU CAN ONLY USE FULL SIZE PLASTIC SPOOLS

Set the spool in the holder with the filament rolling over the top of the spool coming towards you.

Push the gray tab towards the spool and insert the filament into its hole. The filament will automatically pull itself in and out a couple times.



On the screen, select the spool icon on the left sidebar. Change each filament color and type by selecting the corresponding slot's Edit button.

Choose the brand (or Generic if not listed) and the filament type (either PLA, PLA Silk, or PETG). Then choose the color, and finally Confirm.



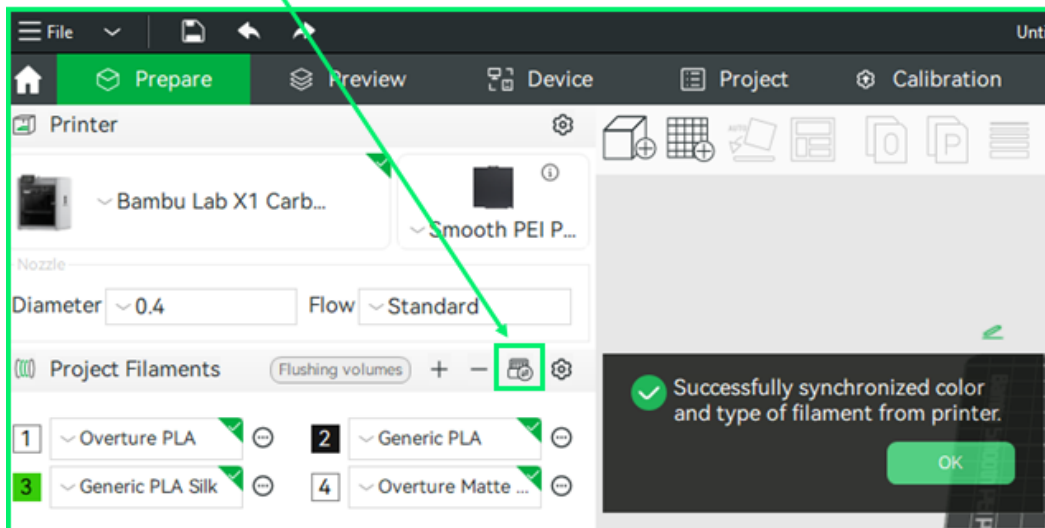
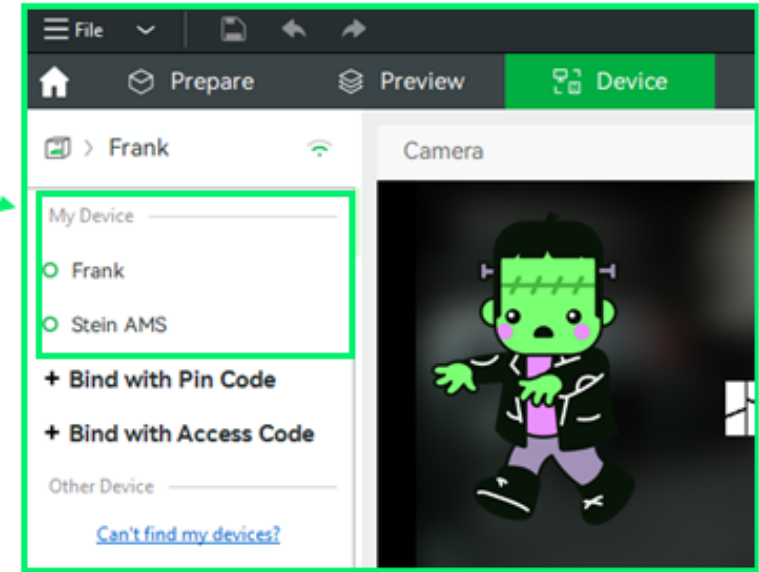
AMS Bambu - Sync Filament



In Bambu Studio, choose Device, then make sure you have the AMS printer selected: **Stein AMS**.

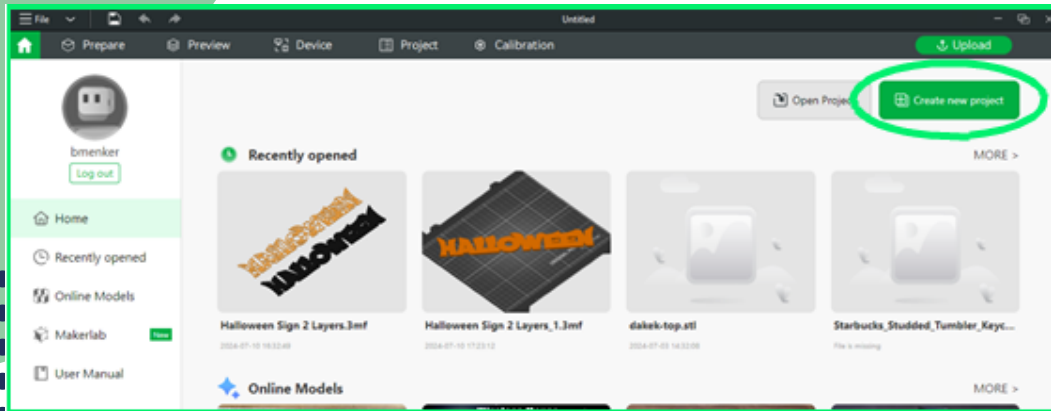
Select the Prepare tab to go back to the main page in Bambu Studio. Here, you'll select Synchronize AMS.

This will sync the colors in Bambu Studio that you chose earlier on the machine.

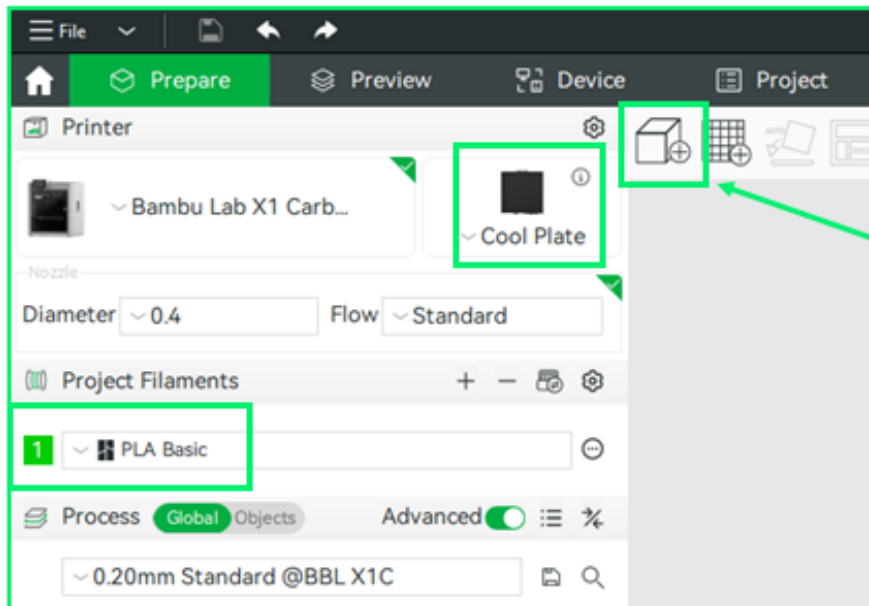


Bambu – Load the File

Open the Bambu Studio software & select “Create New Project”.



Ensure the plate type and filament type are set in the software for the material you are using. Most print jobs will be with **PLA filament** and the **Cool Plate**.

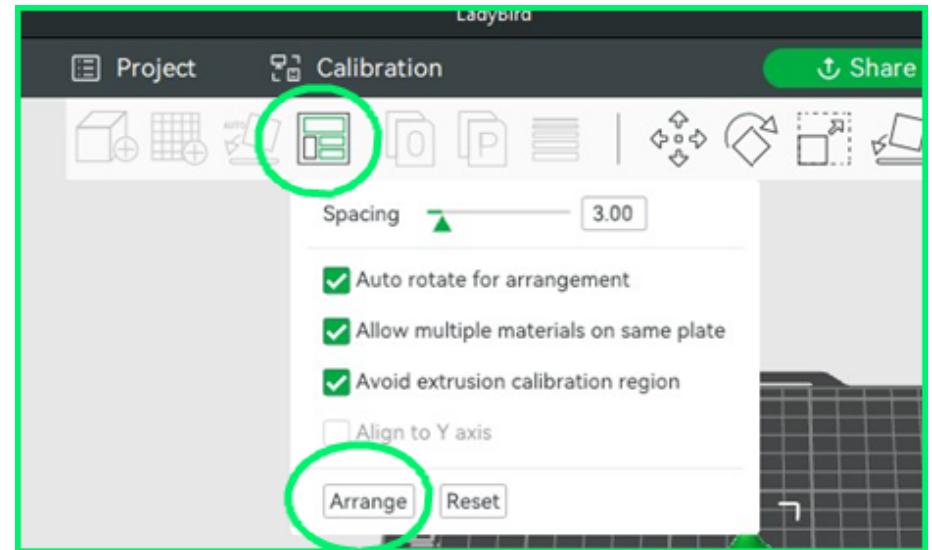


Select the cube with the “+” sign to open your file.

Bambu – Add Supports

Click on the “**Arrange**” function on the top banner & make the spacing 3.00 mm. This will automatically arrange all of your designs on the print bed as close as possible for better printing time. Click “**Arrange**”.

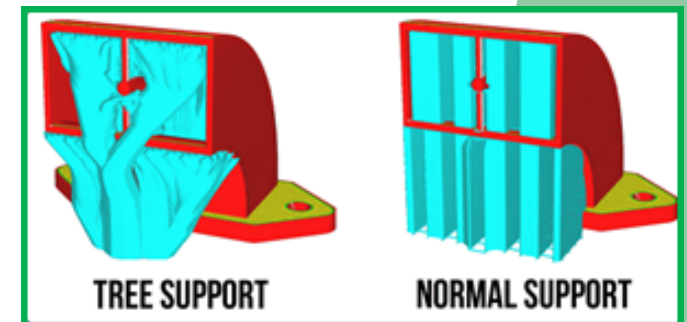
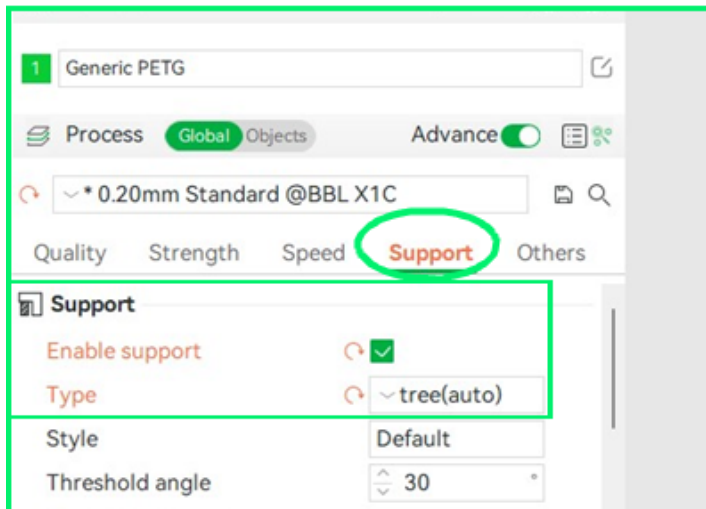
**Your designs may need to be arranged again if large supports are needed.



Next, you may need to add supports to your design.

Go to the supports tab & checkmark the box for “**Enable Support.**” There are 2 support types to choose from: **normal** or **tree**.

Support type choice will depend on your preference and the type best for your design.



Bambu - Toolbar Settings



1. Add Object: Allows you to import an object on the plate
2. Add Build Plate: Allows you to add another build plate, if desired
3. Auto Orientation: Automatically places the object in a particular orientation
4. Auto Arrange: Automatically arranges object on the build plate
5. Split into objects: Splits model typically when using Text Tool
6. Split into parts: Splits model when using the Assemble Tool
7. Variable layer height: Creates different layer heights for faster printing
8. Move: Translates the model in the X, Y, and Z directions
9. Rotate: Rotates the model about a selected internal axis by dragging the rings around the object
10. Scale: Scales the model in the X, Y, and Z dimensions in mm or by a percentage
11. Lay on Face: Adjusts the nearest flat edge of the object onto the platform
12. Cut: Cuts the object in to the amount of parts desired and the location selected
13. Mesh Boolean - make Union, Difference (Subtraction), or Intersection between two parts
14. Support Painting - Manually add supports
15. Seam Painting - Change/move seams
16. Text Shape - Insert layers of text
17. Color Painting - Select parts of your object to print in different colors (AMS required)
18. Measure - Measure dimensions of an individual part
19. Assemble - View model fully assembled
20. Brim Ears - Adds brim to corners of model to prevent warping
21. Assembly View - Allows you to view the model fully assembled

Bambu - Slice Settings



Print Settings

1. Quality - layer height, line width, seam, precision, walls
2. Strength - walls, top/bottom shells, infill,
3. Speed - layers, travel, acceleration
4. Support - enable support, raft
5. Others - bed adhesion, prime tower, flush, special modes

Filament Settings

1. Filament - Adjust Nozzle Temperature
2. Cooling - Cooling Fan Settings
3. Setting Overrides - Retraction Settings
4. Advanced - G-code
5. Notes - Input Notes

The screenshot shows the 'Print Settings' window in Bambu Studio. At the top, there are five tabs labeled 1 through 5: Quality, Strength, Speed, Support, and Others. The 'Quality' tab is selected. Below the tabs, there are two main sections: 'Layer height' and 'Line width'. Each section contains several settings with input fields and units.

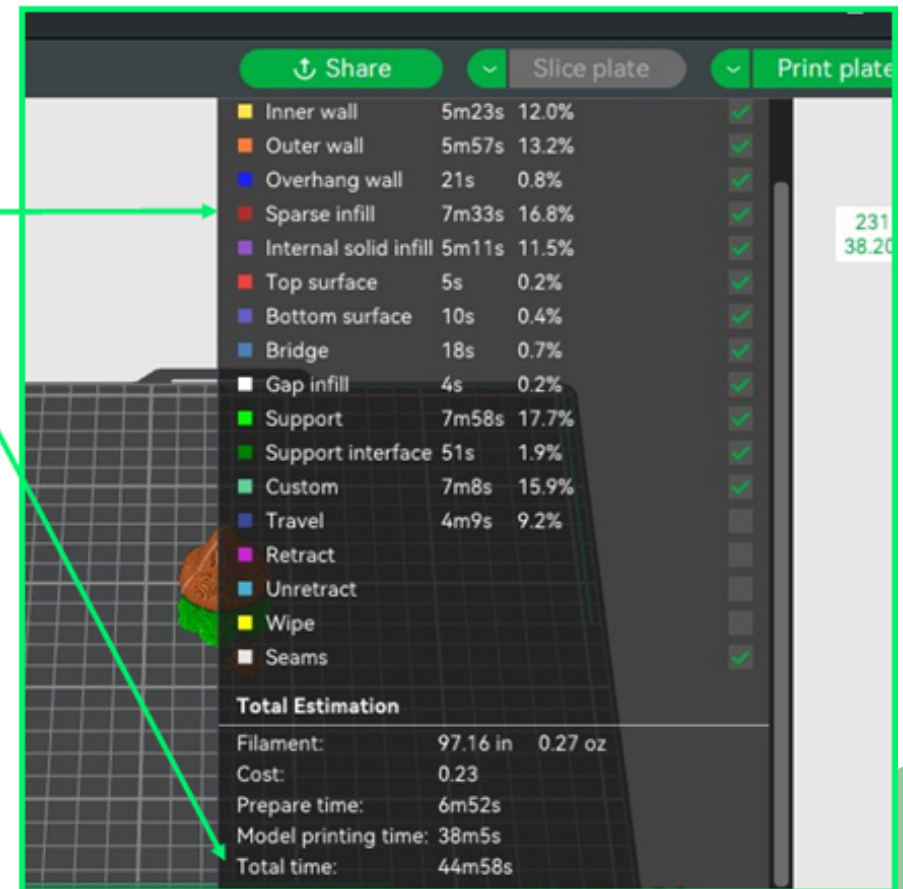
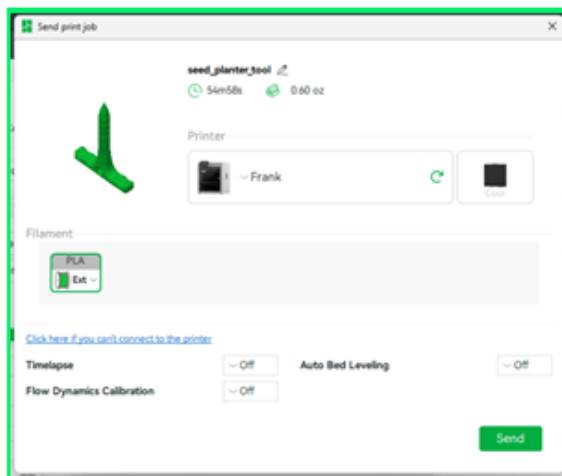
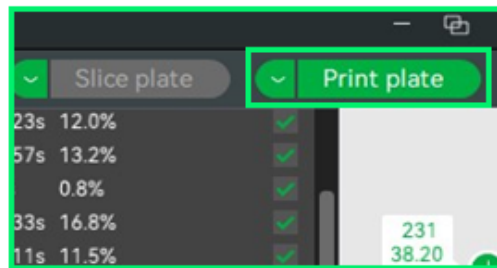
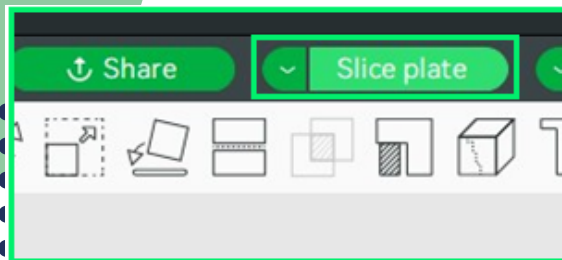
Section	Setting	Value	Unit
Layer height	Layer height	0.2	mm
	Initial layer height	0.2	mm
Line width	Default	0.42	mm
	Initial layer	0.5	mm
	Outer wall	0.42	mm
	Inner wall	0.45	mm
	Top surface	0.42	mm
	Sparse infill	0.45	mm

The screenshot shows the 'Filament settings' window in Bambu Studio. At the top, there is a 'Project Filaments' section with a dropdown menu showing 'PLA Basic'. Below this, there is a 'Filament settings' section with a dropdown menu showing 'Generic PLA'. The 'Advanced' toggle is turned on. Below the 'Advanced' toggle, there are five tabs labeled 1 through 5: Basic information, Print temperature, Cooling, Retraction, and G-code. The 'Basic information' tab is selected. Below the tabs, there are several settings with input fields and units.

Section	Setting	Value	Unit
Basic information	Type	PLA	
	Vendor	Generic	
	Default color		
	Diameter	1.75	mm
	Flow ratio	0.98	
	Density	1.24	g/cm³
	Price	20	money/kg
	Softening temperature	45	°C
	Recommended nozzle temperature	Min 190, Max 240	°C
	Print temperature	Nozzle Initial layer	220
Nozzle Other layers		220	°C
Cool Plate / PLA Plate Initial layer		35	°C
Cool Plate / PLA Plate Other layers		35	°C
Engineering Plate Initial layer		0	°C
Engineering Plate Other layers		0	°C

Bambu – Slicing

Slice the plate. This may take a minute depending on how complex your design is. Once slicing is complete, a report will show how long it will take to print as well as a print time breakdown for each part of the design.



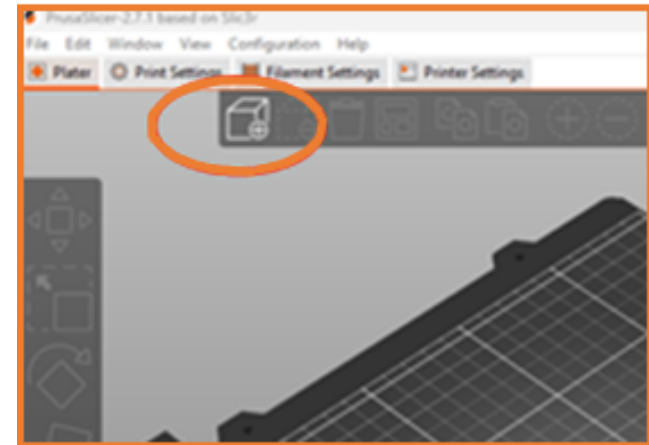
Once you are happy with the print job & time, click **“Print Plate.”** Check your device, plate, and filament in the pop up, then click **“Send.”**

The Bambu is able to print wirelessly so after a few seconds, you should see the Bambu light up and the design on the mini home screen on the 3D printer itself.

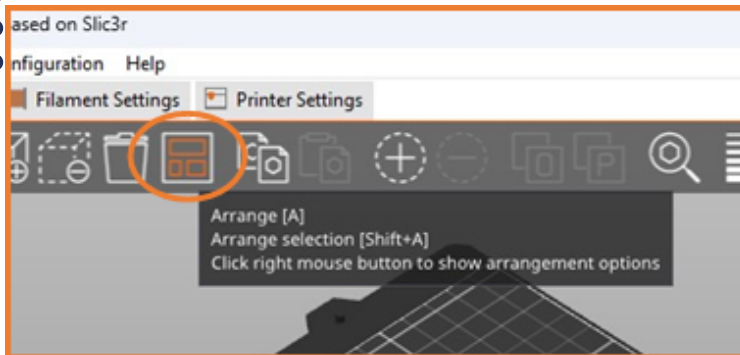
Prusa - Load the File



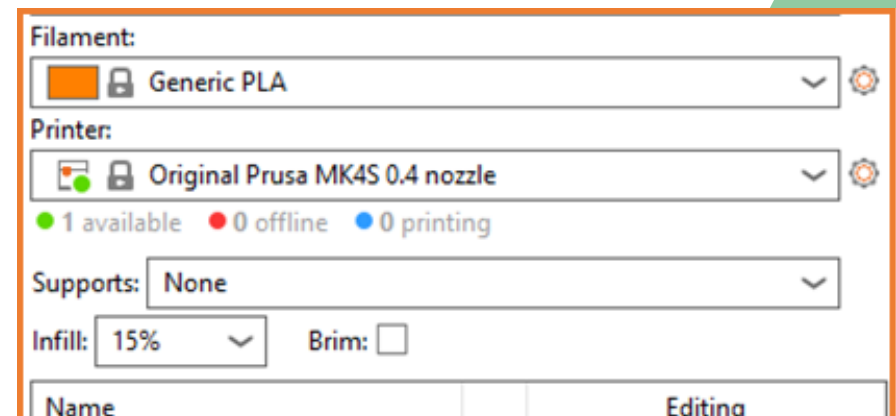
Open the PrusaSlicer software. On the top banner, select the cube with the + sign on the far left. Once you click on the cube, a window opens where you can select your file.



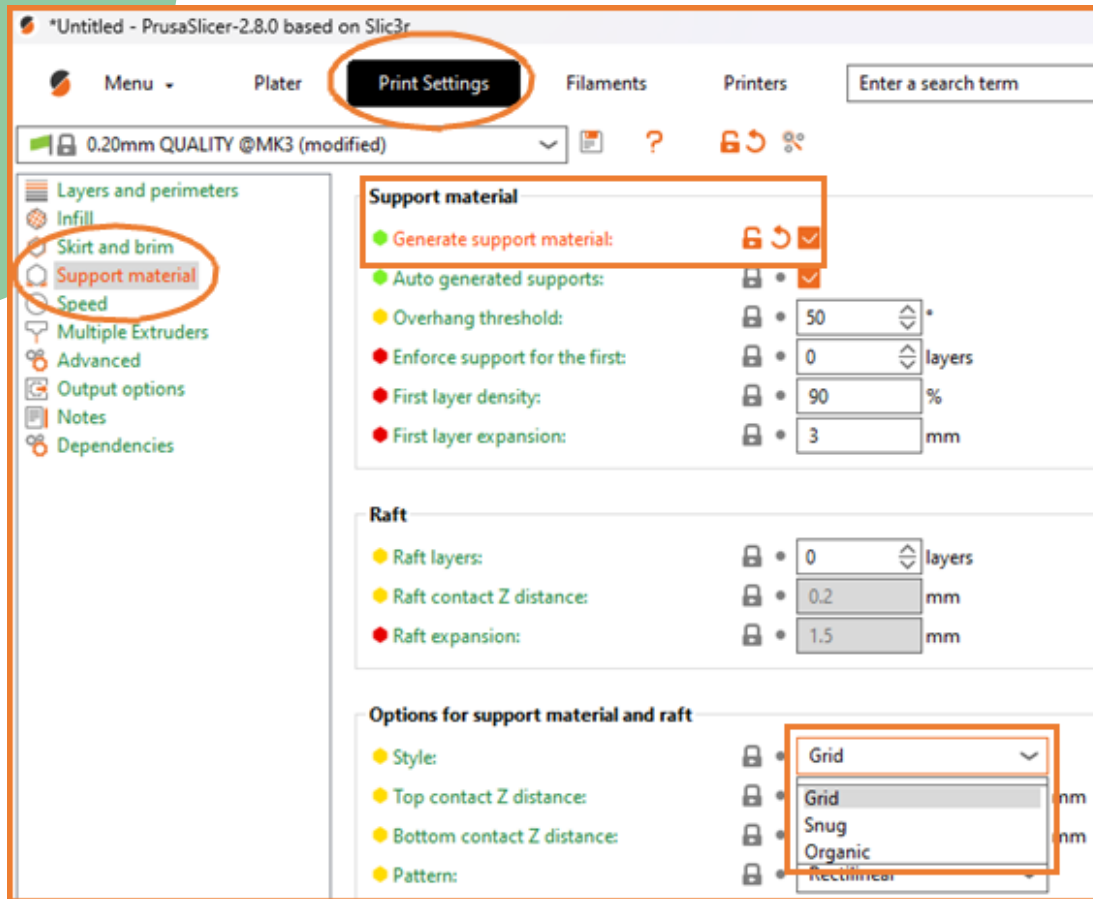
Select “**Arrange**” on the top toolbar & this will automatically arrange all of your designs on the print bed as close as possible for a quicker printing time.



Ensure the filament type is set in the software for the material you are using. Most print jobs will be with **PLA filament**. The print setting “**0.20mm QUALITY**” and infill setting of **15%** is a good default setting, but can be modified for prints with finer details. Checkmark “**Brim**” if your print needs more surface area on the plate to prevent it from moving.



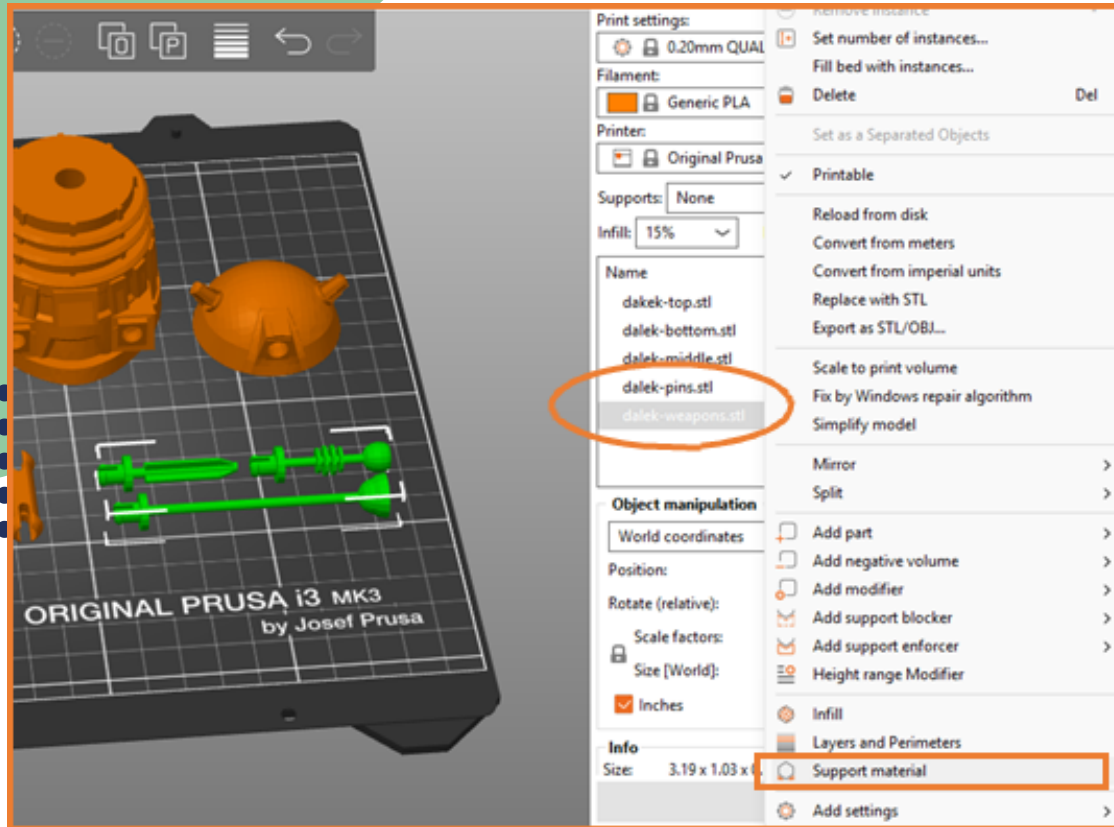
Prusa - Add Supports



The type of supports can be changed under **“Print Settings”** then **“Support Material.”** Checkmark **“Generate Support Material”** and near the bottom, you can choose which type: Grid, Snug, or Organic.

This will automatically enable supports for any overhangs in your design.

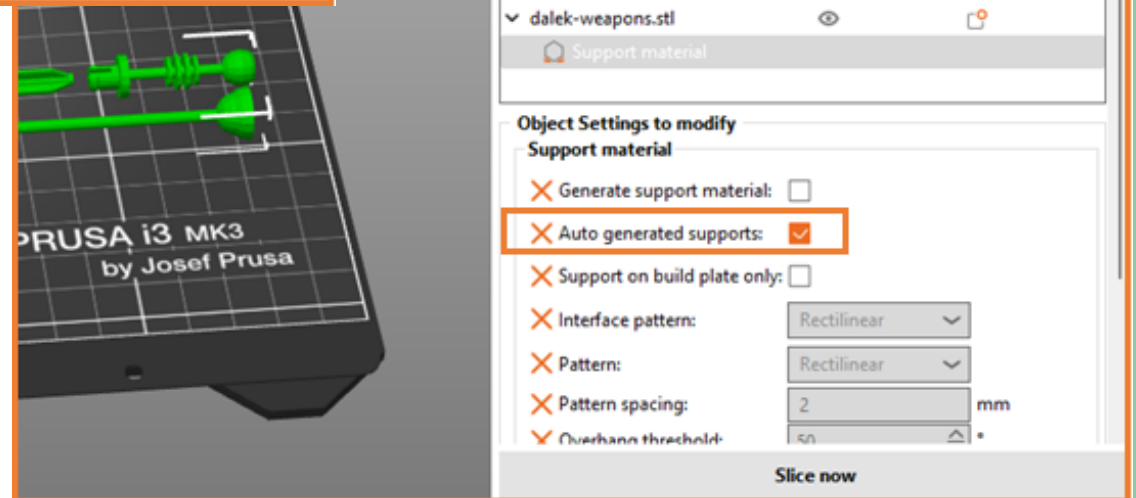
Prusa - Supports



If you have added multiple files, but not all of the files need supports, you can remove supports from those files while still leaving supports on other files.

On the right-hand side of the screen lists your files. On the file you do not want supports generated, right click, then select **“Support Material.”**

This brings up a menu that allows you to deselect **“Auto Generated Supports.”**



Prusa - Unload/Load Filament



To unload filament, use the knob or touch the screen for **Filament**, then **Unload**.

To load, insert the filament into the extruder.

The Prusa will automatically pull it in.

The Prusa will ask what type of filament is being loaded. Choose the correct option. Then, it will ask if the filament extruding is the correct color.

Continue to press No until it is the correct color, then press **Yes**.



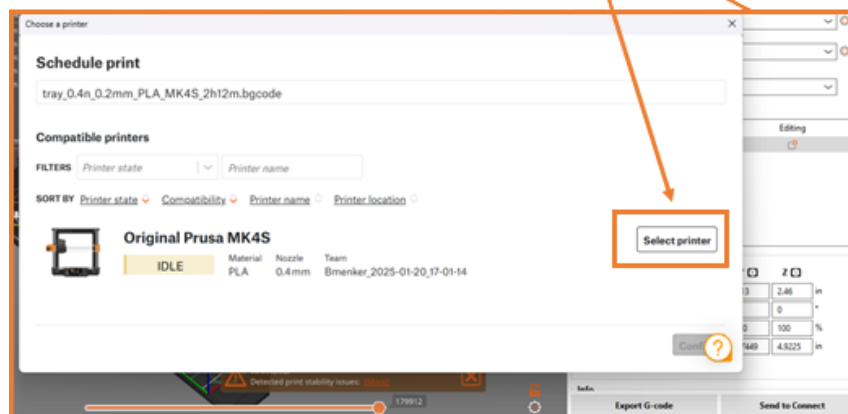
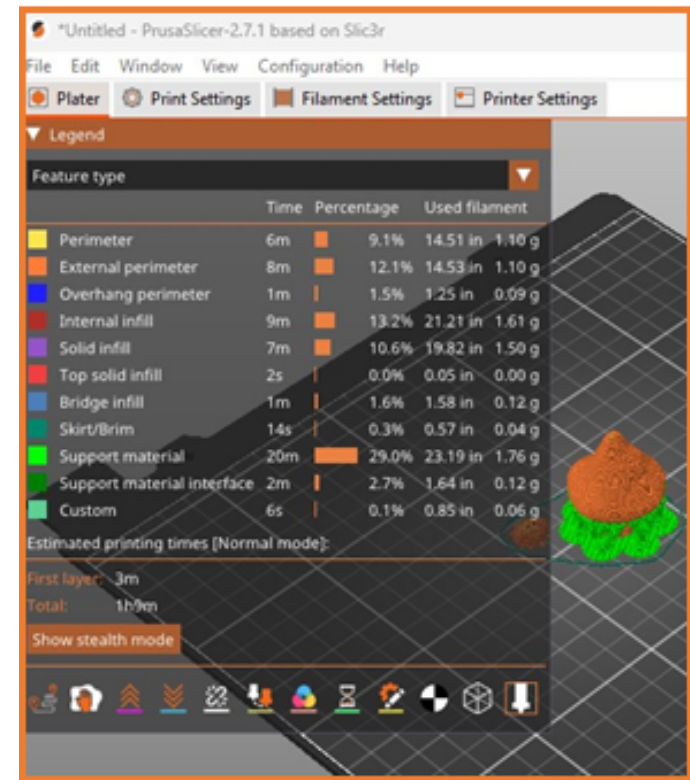
Prusa - Slicing



Once you are satisfied with your design, click **"Slice Now"** to begin creating the G-Code. This may take a minute depending on how complex your design is. Once slicing is complete, a report will show how long it will take to print as well as a print time breakdown for each part of the design.

Once you are happy with the print job & time, click **"Send to Connect"** in the bottom right-hand corner.

A page will appear asking you to select our printer. Once selected, choose the option **Set ready and print now**, then **Confirm**.



 **Set ready and print now**
Is the print sheet empty and clean?