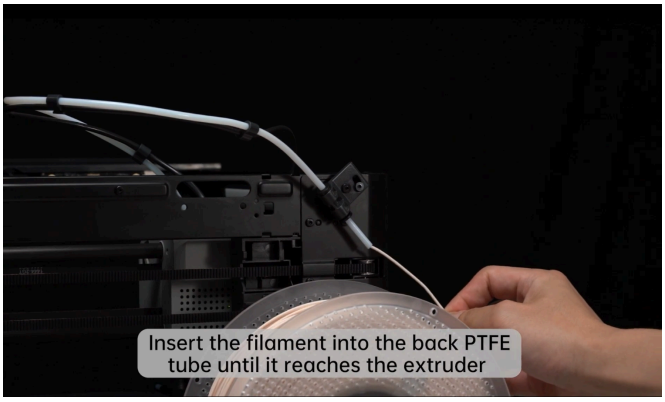
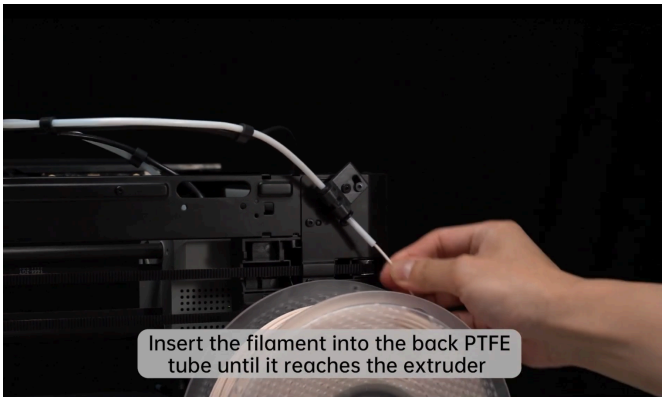
Bambu Lab P1P

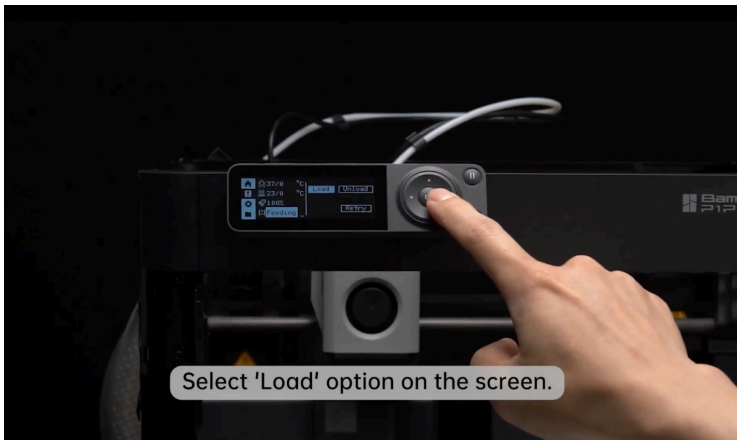
April 2024

1. Insert the filament into the back PTFE tube until it reaches the extruder.

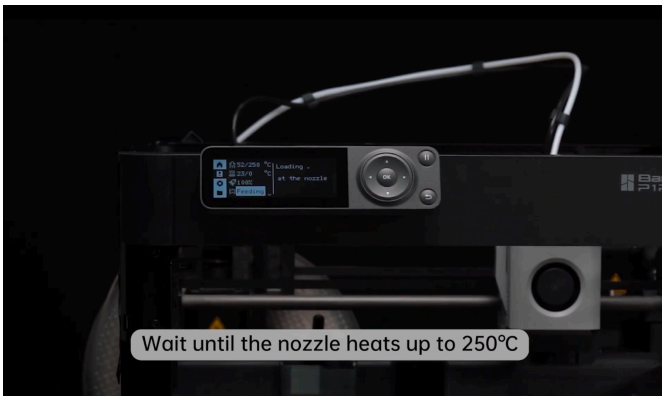


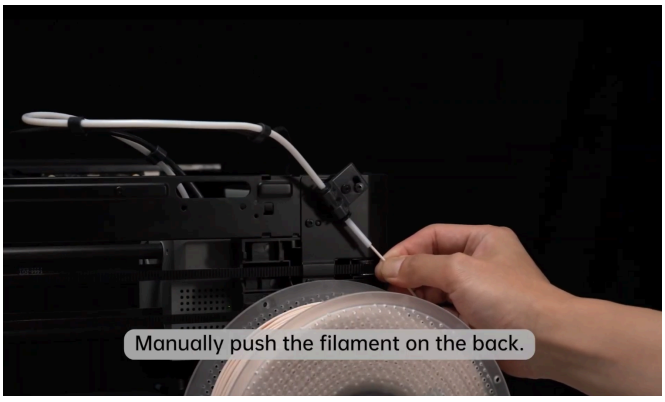


2. Select the 'Load' option on the screen.

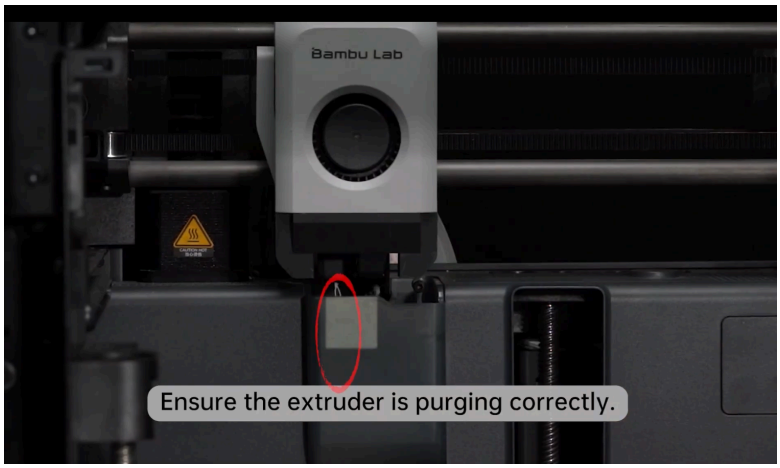


3. Wait till the nozzle heats up to 250 ℃; Manually push the filament on the back.



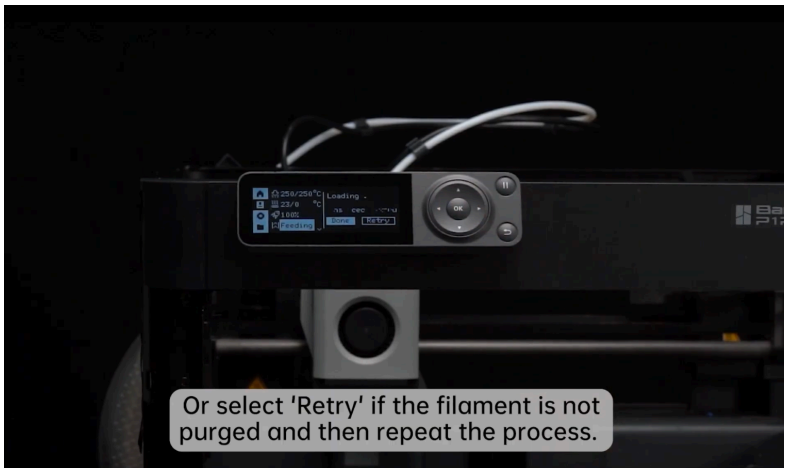


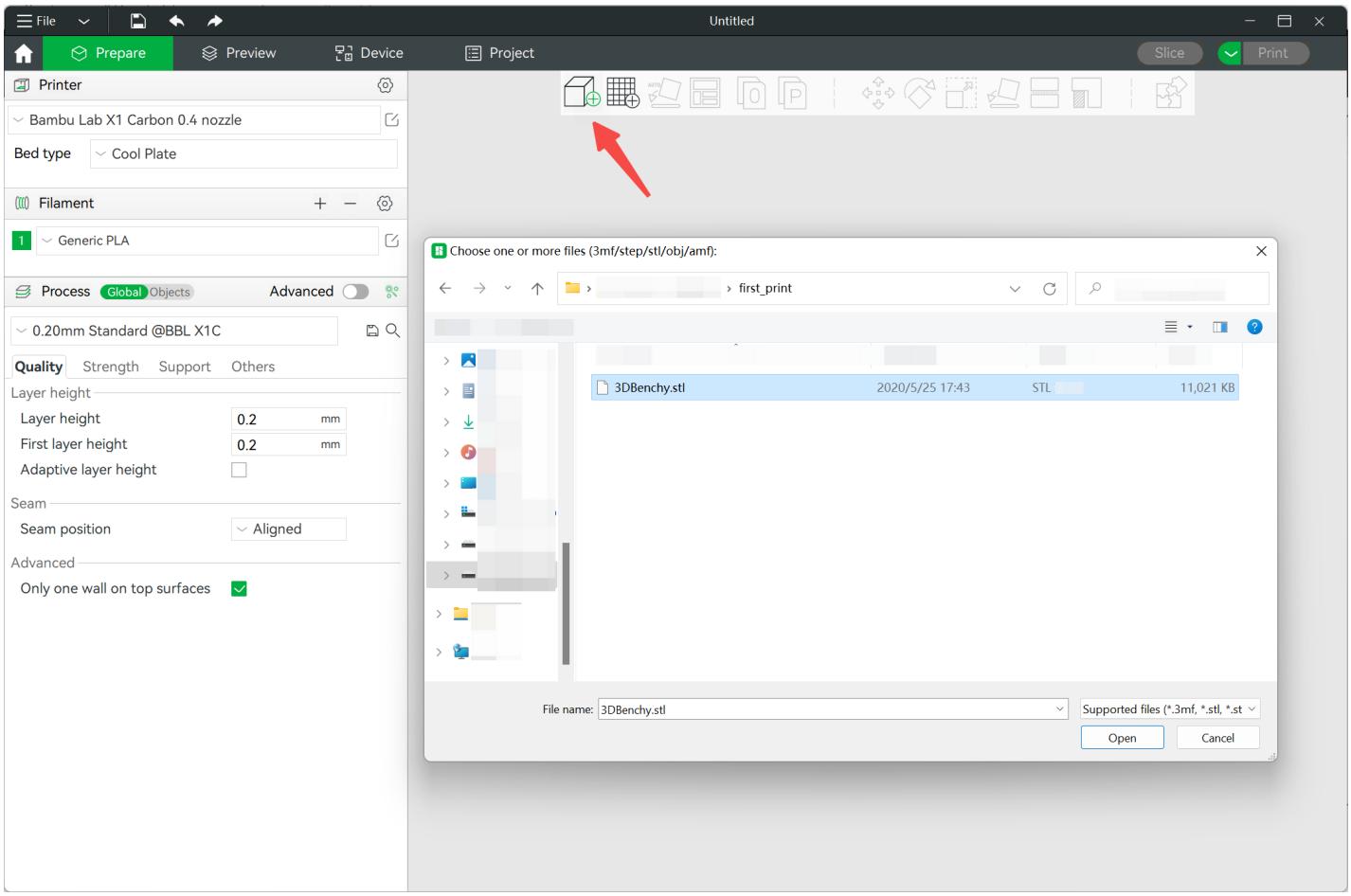
4. Ensure the extruder is purging correctly.



5. Select 'Done' to finish.

6. Or select 'Retry" if the filament is not purged and repeat the process.



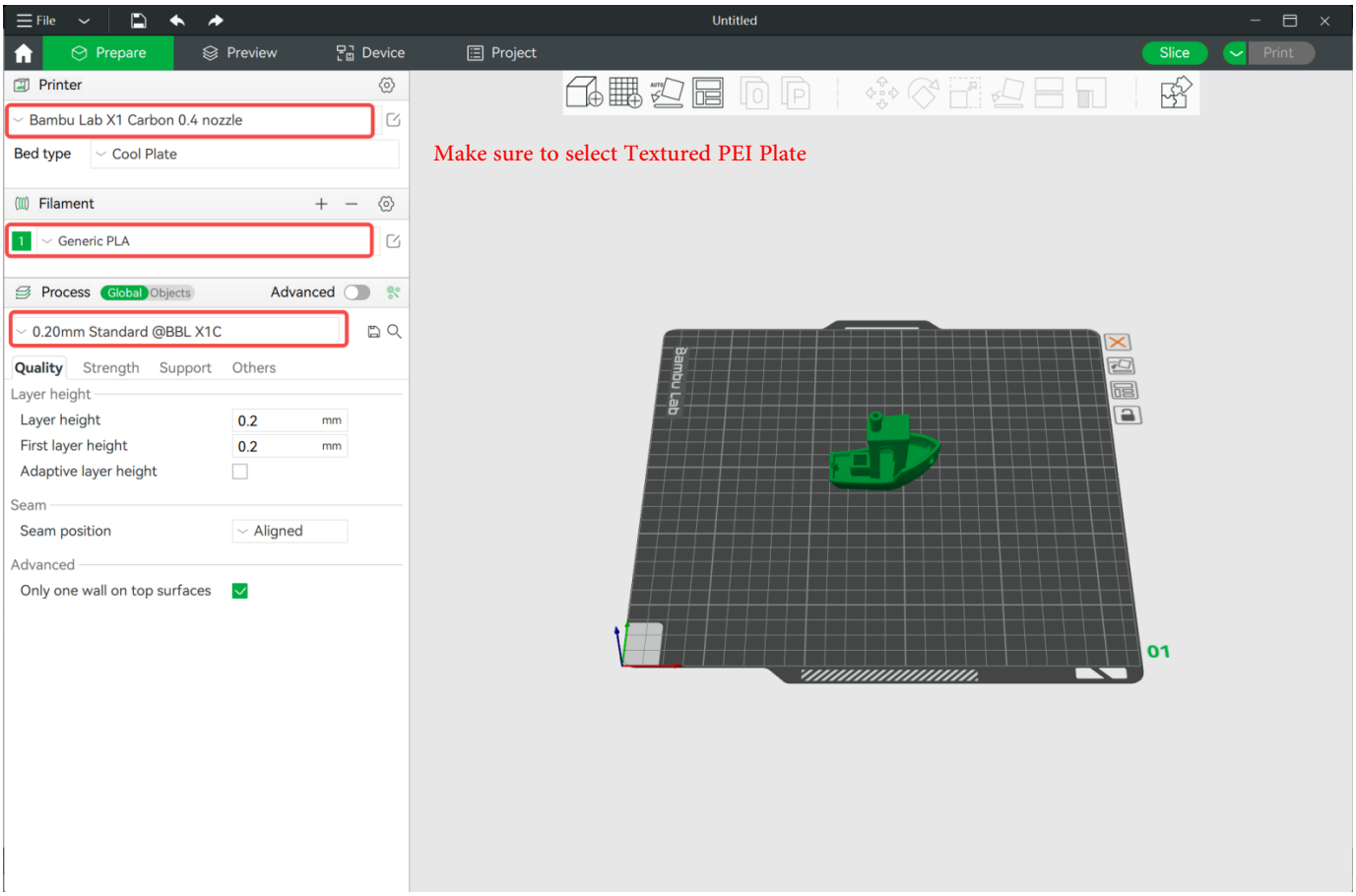


Select Printer/Filament/Process presets

To start slicing the model, you need to choose the presets for the machine you are using, for the filament you will print with and also the settings you want to print the model in.

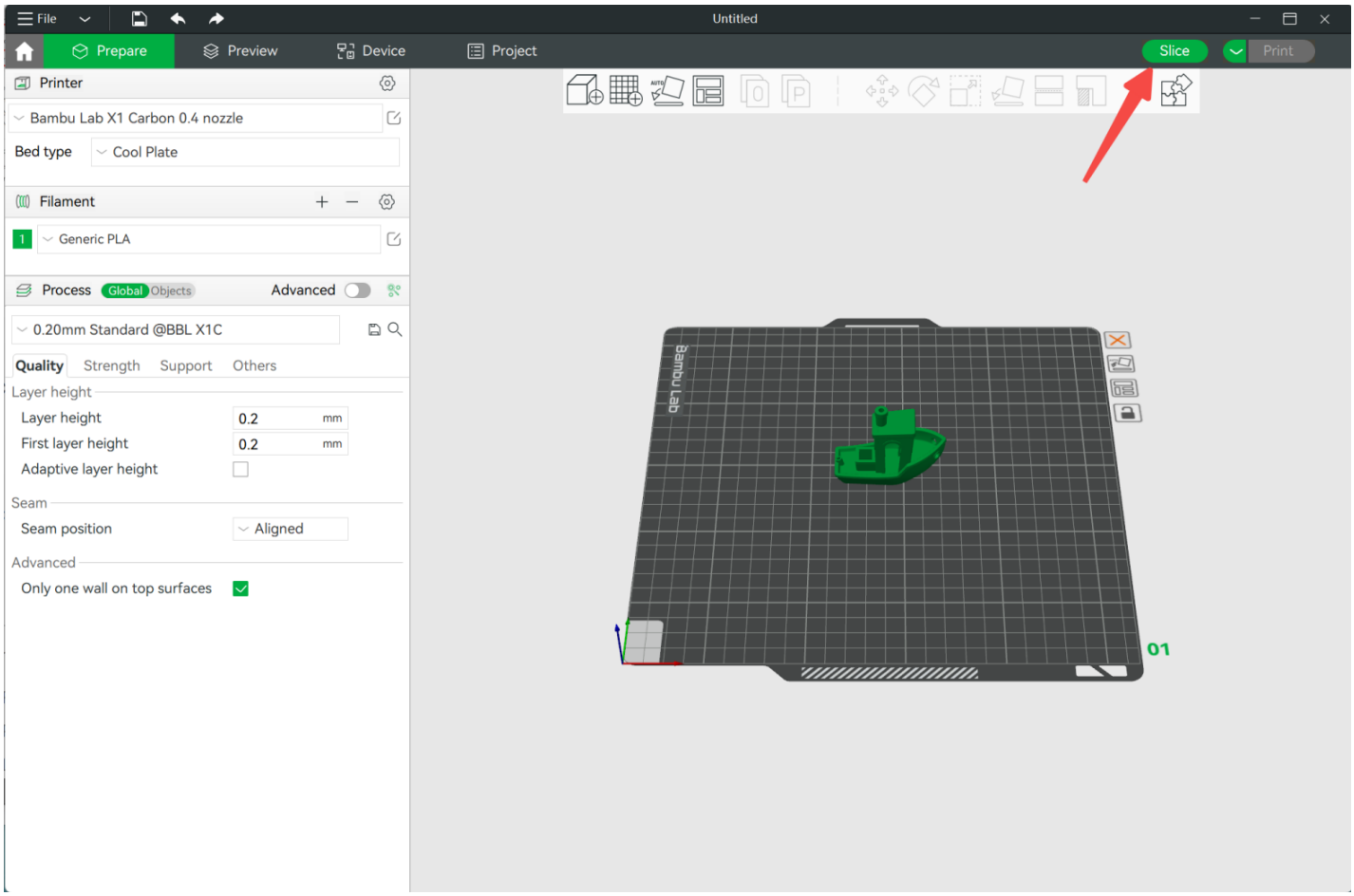
1. Select the printer you are using from the drop-down list under “Printer”. This will also include the nozzle size you will be printing with
2. Under the "Filament" section, select the type of filament you intend to use from the drop-down list
3. Finally, choose the layer height you want your model to be printed in from the “Process” drop-down menu. Always remember that the smaller the layer height, the longer the print will take. For the majority of prints, a 0.20mm layer height is the norm.

Make Sure to select P1P 0.4 Nozzle Under Printer Select, and set Bed Type to Textured PEI Sheet

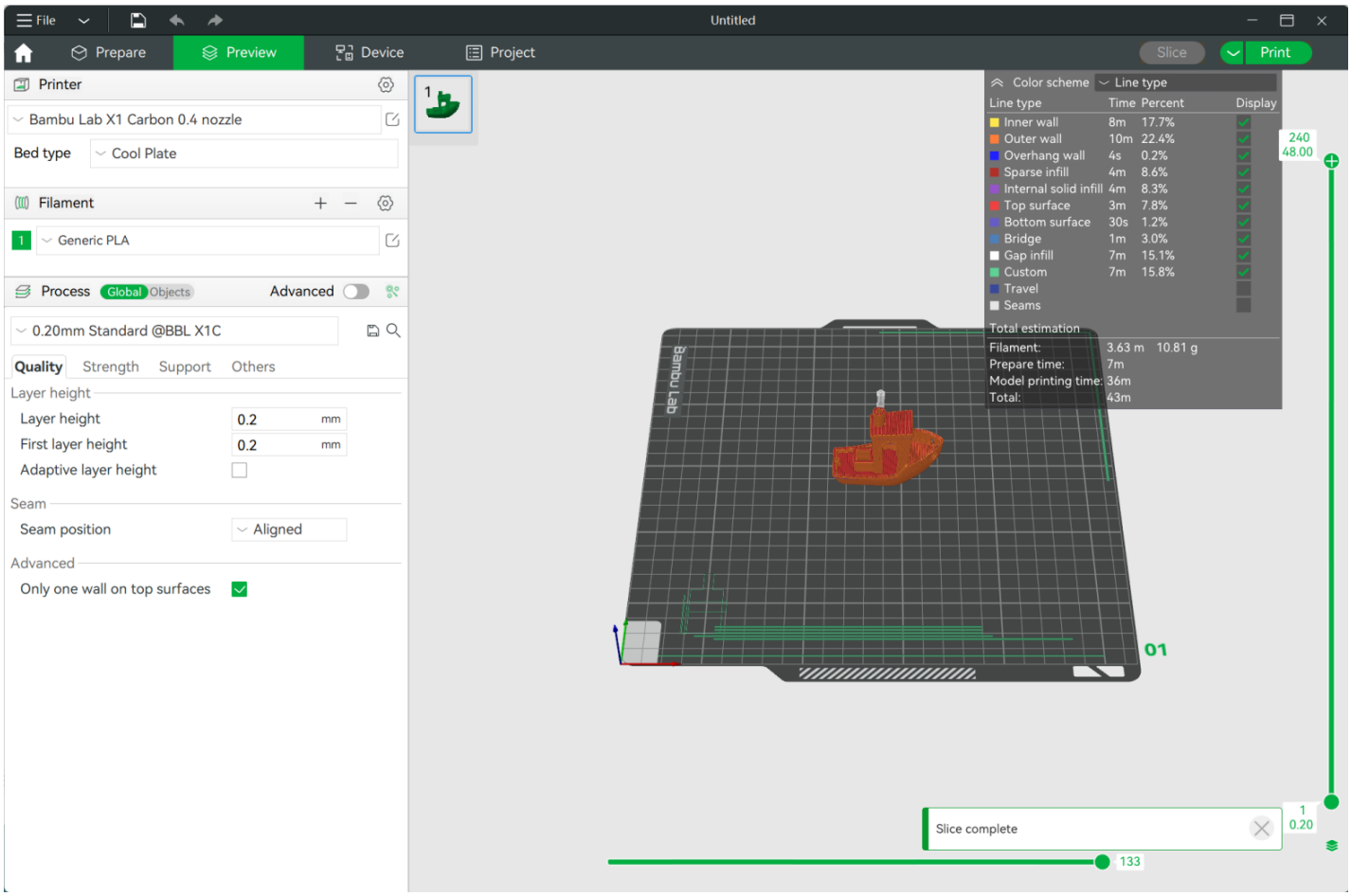


Click the "Slice" button

Once done, click on the “Slice” button located on the top hand right of Bambu Studio. This will generate a .3mf file which is the file format used for the printer to be able to print the model.



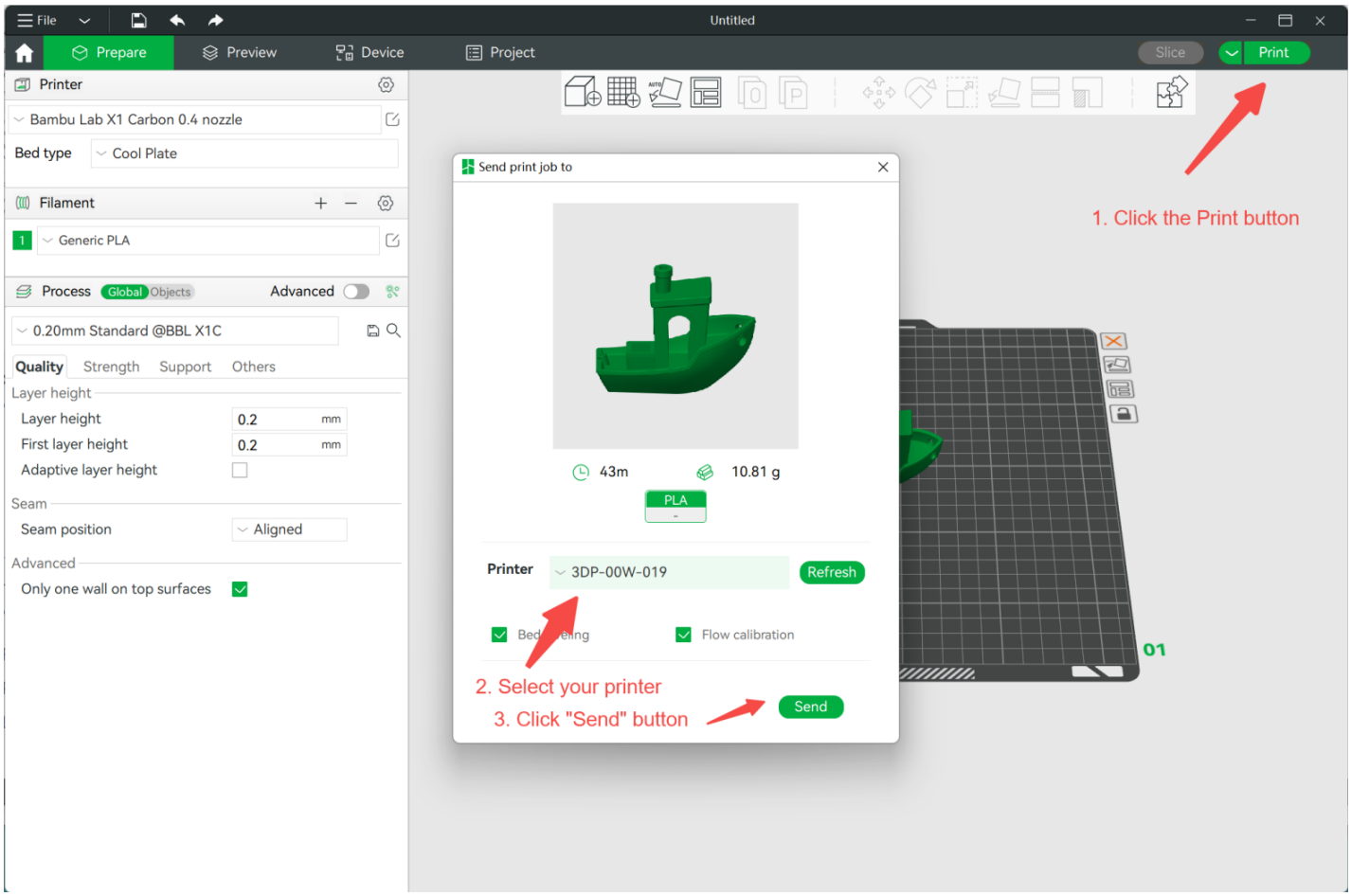
Once done, the slicer will take you to the Preview pane which will show you what the sliced model looks after processing the .3mf file. The histogram on the right hand side will also show you information on the printing times for each parameter of the print.



Send print job via WLAN or SD Card

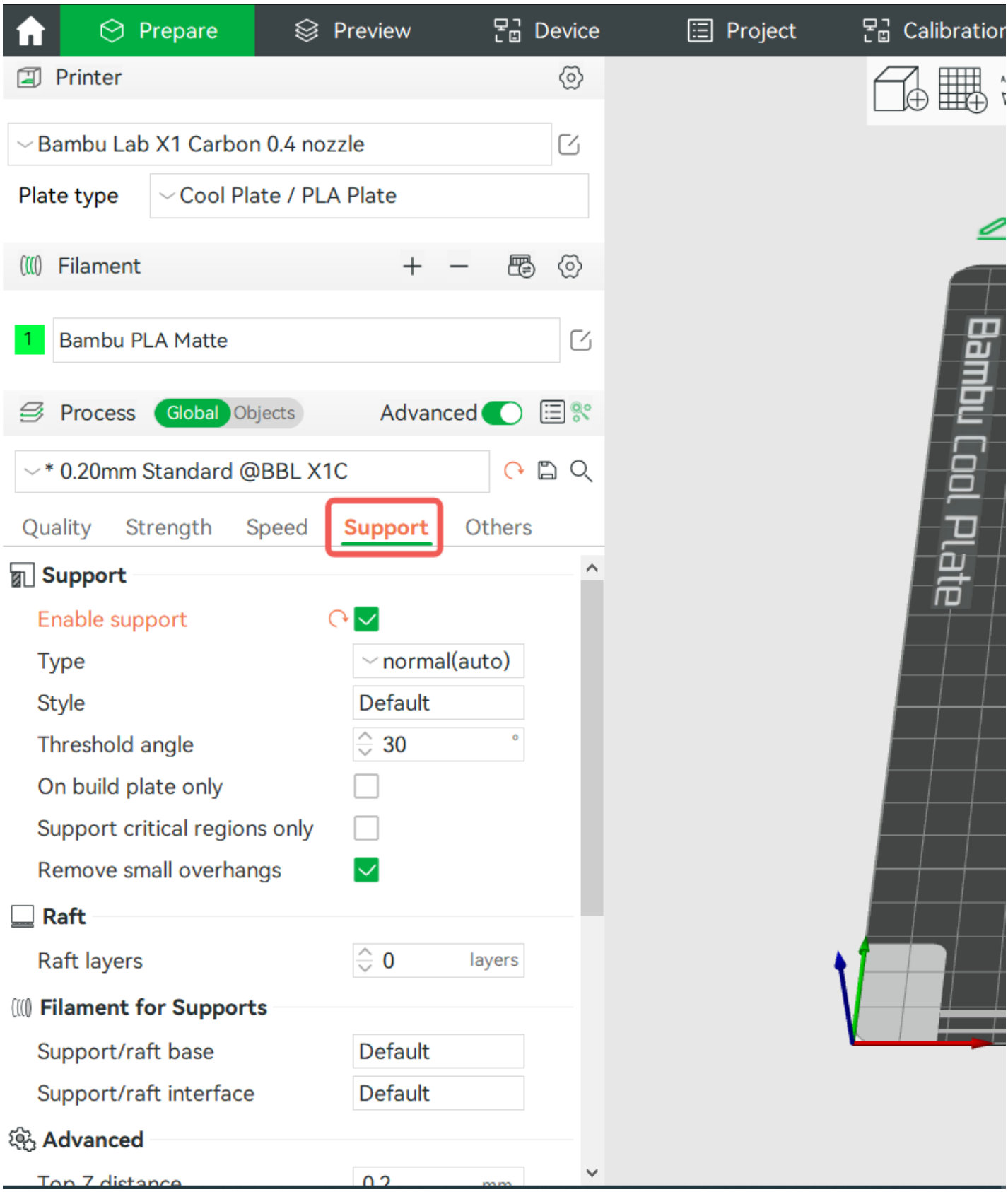
To send the print job to the printer via WLAN, click “Print” on the top right-hand corner. This will prompt a pop-up window with a quick preview of the model and will also ask you to select the Printer you want to send it to from the drop-down list, and you will also give you the option to choose whether or not you want the printer to perform certain functions like Bed Leveling, flow calibration, etc before the print starts. Once done, click “Send” to send the file to the printer and start printing

Note: *You will need to have Bambu Network plug-in installed to be able to send files via WLAN*



Export the sliced file to SD card and print offline

To use the SD card file transfer option, click on the down arrow next to the “Print” icon on the top right and select “Export Sliced File”. Once done, The “Print” icon will change to “Export Sliced File”; click on it. A file explorer window will pop-up in order for you to select the location of the SD card. nce located, click on “Save” and the file



Support types

There are 2 basic types of support: normal and tree. The main difference between the two types is:

- **normal support** directly **projects the overhangs down to the heat bed, and gets the support body;**

- **tree support** samples the overhangs to get the so-called `nodes`, each node is represented as a circle. And then the nodes are propagated down to the heat bed. During propagation, the circles may be enlarged to get better strength and may be moved away from the object so the supports are less likely to collide with the object.

On the support page, we can select 5 types of support, which are variants or combinations of these two types:

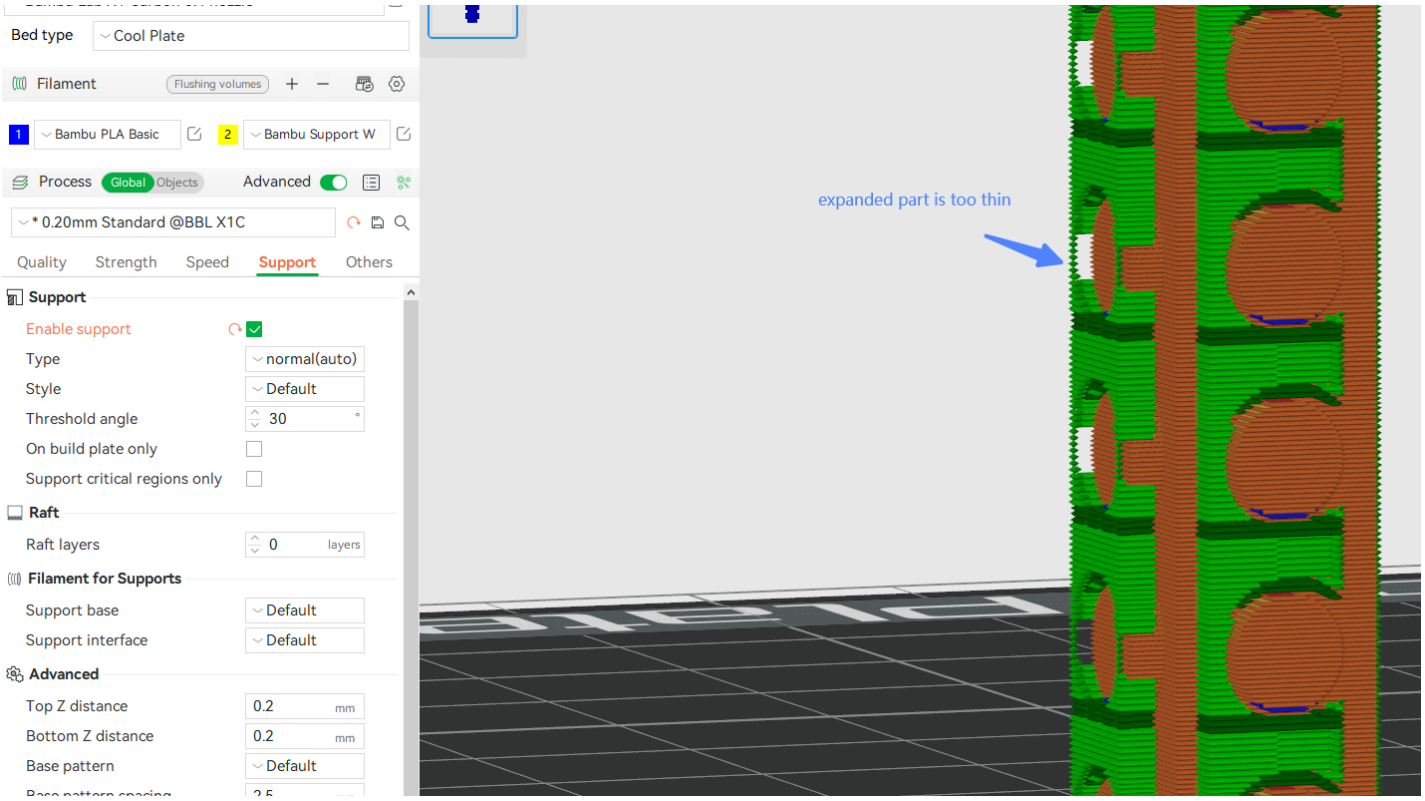
1. `normal(auto)` : normal support with automatically detected overhangs.
2. `tree(auto)` : tree support with automatically detected overhangs.
3. ~~`hybrid(auto)` : combination of `normal(auto)` and `tree(auto)`, that is, when the overhang area is large, use `normal(auto)`, otherwise use `tree(auto)`.~~
   After version 1.4.1, we moved hybrid(auto) from type to style. To enable it, please select type=tree(auto) and style=Tree Tybrid.
   We made this change because we added a new style (tree slim), and possibly we'll add more styles. It's not appropriate to use support types to do this, or we'll have too many support types. But in fact, tree slim, tree strong, and tree hybrid are only different in some parameters. They are all tree support essentially.
4. `normal` : generate normal support only on support enforcers.
5. `tree` : generate tree supports only on support enforcers.

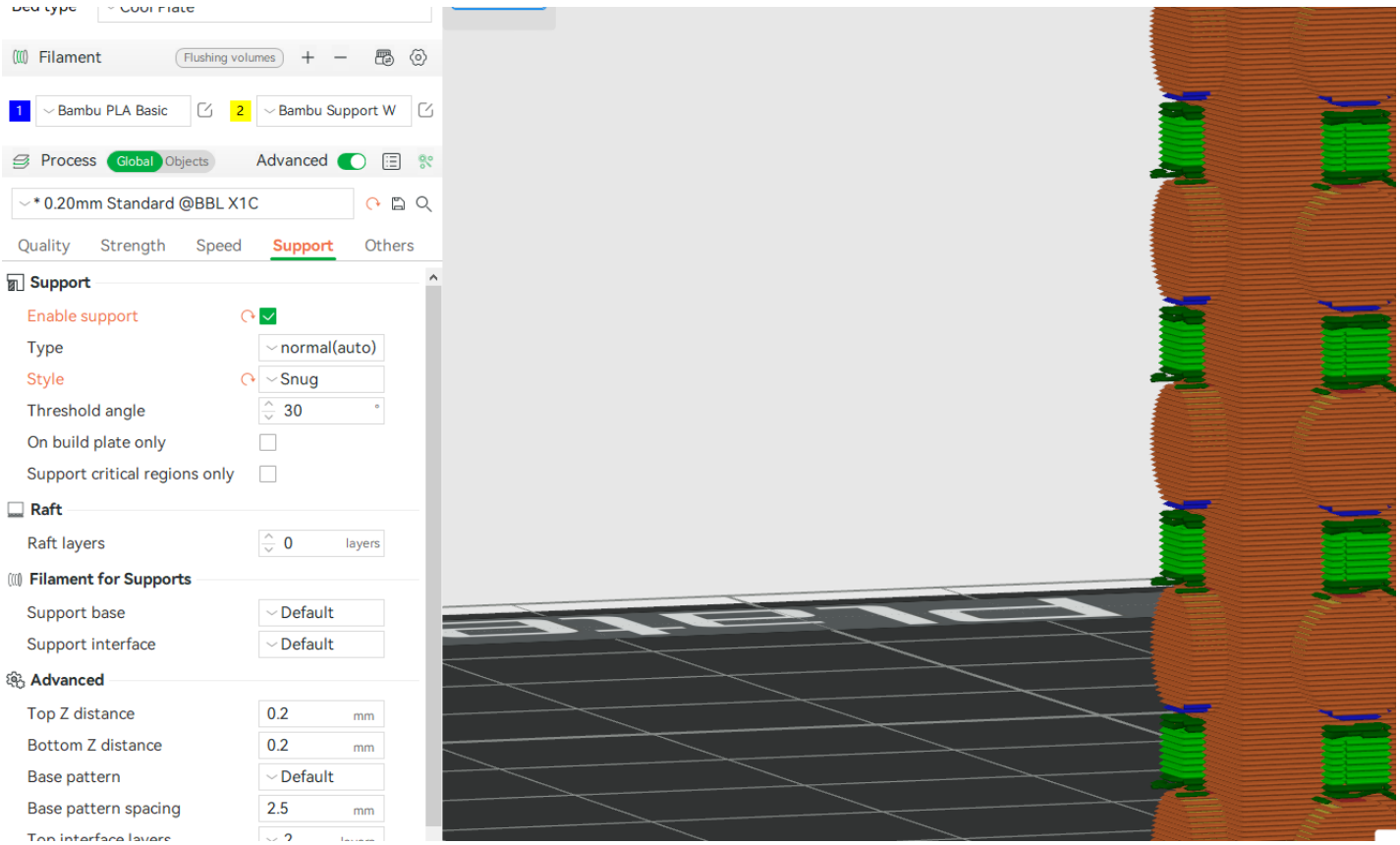
Support styles

Both normal and tree supports have different styles to further adjust the final support structure.

Normal support has two styles:

- **Grid**: the support region is expanded and normalized to rectangles. This is the default style of normal support.
- **Snug**: the support region is NOT expanded, but tightly aligned with the overhang areas. This style is useful when the expanded supports have any side effects, such as in the following case.





Tree support has three styles:

- **Tree Slim**: this features an aggressive branch-merging strategy. As a result, a much smaller support volume is generated without sacrificing strength (by automatically increasing the wall count and using smoother branches).

- **Tree Strong**: the old style that is strong, but sometimes difficult to remove.
- **Tree Hybrid**: **the current default style,** which is the hybrid of tree strong and normal grid. Below the big flat overhang regions, normal grid supports are generated. Otherwise, it will generate the tree strong supports.

Common options

Threshold angle

The threshold angle is the maximum slope angle that needs support. If a surface's slope angle to the horizon is less than this threshold value, support will be generated when the support type is auto.
The larger this angle is, the more supports will be generated. The default threshold angle is 30 degrees. For most materials, this is a safe angle to print without support.

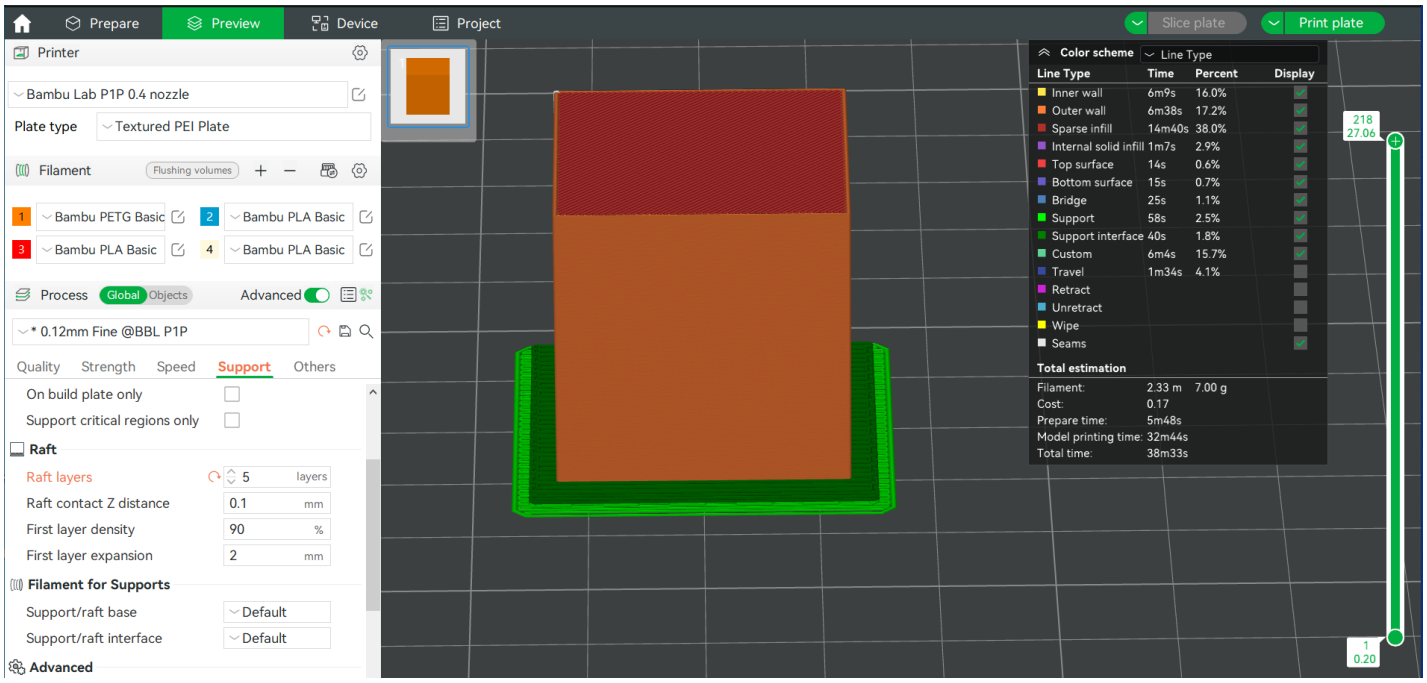
Raft

Raft is a type of support, which is used to generate support at the bottom of the model to lift it up. Usually, when printing materials such as ABS that are prone to warping, then you can enable the raft.

The raft contact Z distance represents the distance between the top of the raft layer and the model.

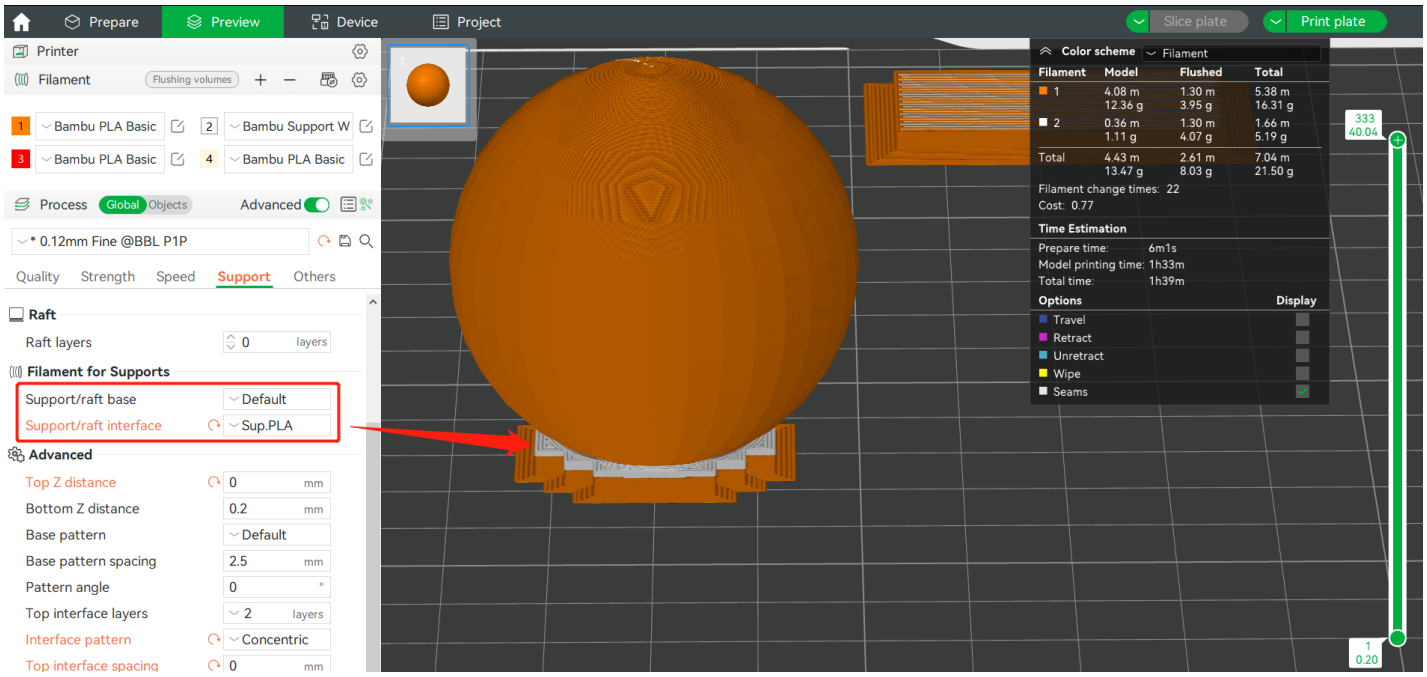
The first layer density means the density of the first layer of the raft and the support.

The first layer expansion can be used to expand the first raft and support layer, improving the bed adhesion.



Support filament

Support is composed of two parts: **base** and **interface**. **Interface** layers are the layers touching the object. The rest of the support body is the **base**. Both parts can use different filaments than the object. Default means no filament is specified and the filament printed at the current layer is used, so filament switching time is minimized. Usually, we select specialized support materials such as support W as the support surface material.

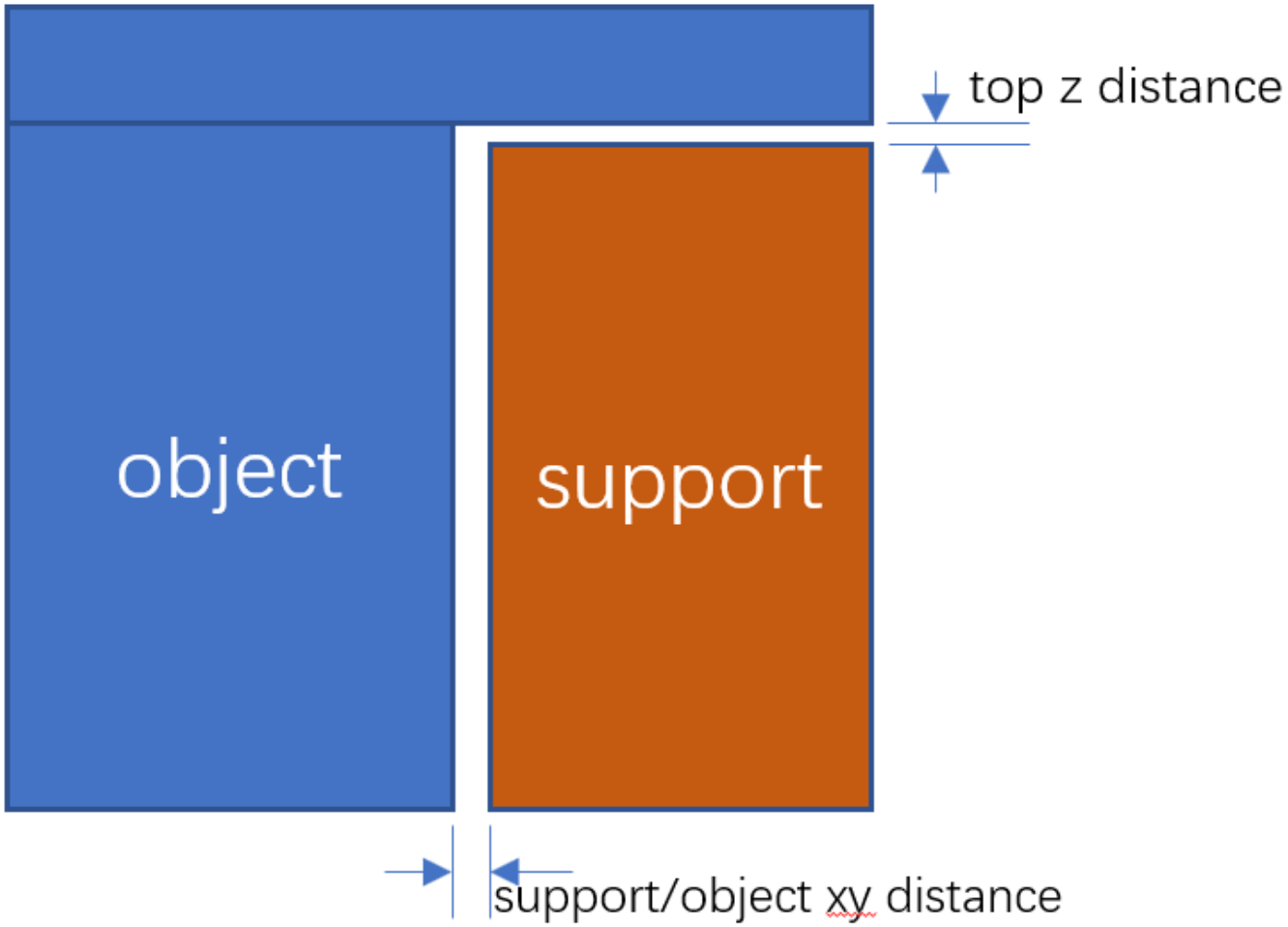


Top z distance

The z distance from the support top to the object, as shown below. When setting to 0, the support filament is assumed to be support material, e.g. Bambu Support W.

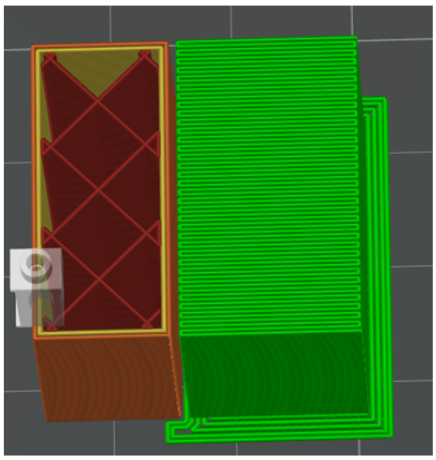
Support/object xy distance

The XY distance between the support and the object, as shown below.

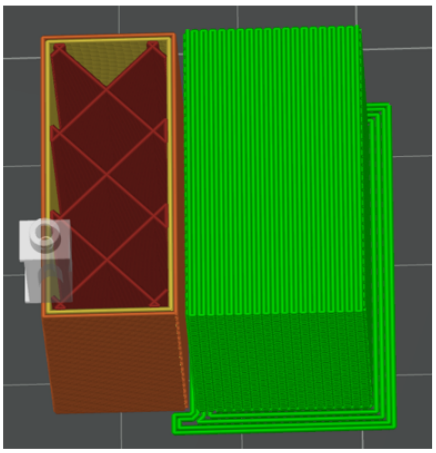


Base & interface pattern settings

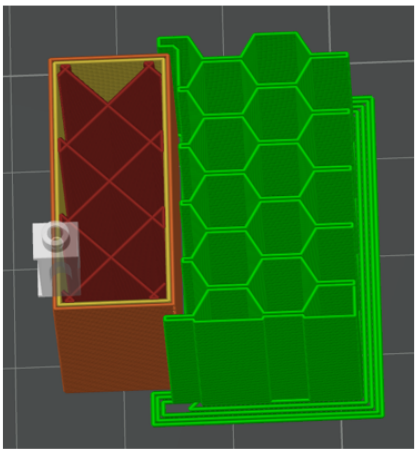
- **Base pattern**
  This is the infill pattern of the support base. There are currently 5 patterns, as shown below.
  - **Rectilinear** is the most commonly used support and **default pattern for normal support,** which usually goes in two directions (left to right, front to back)
  - **Rectilinear grid** is similar to **rectilinear**, except it alternates the direction of every layer, so its strength is much better but can be harder to remove.
  - **Honeycomb** is very different than the other two, and is a good balance of strength and stability for taller support structures.
  - **Lightning** is an extremely sparse infill pattern for tree support, which can save both material and printing time, but with lower strength.
  - **Hollow is a default pattern for tree support**, which means no infill at all.



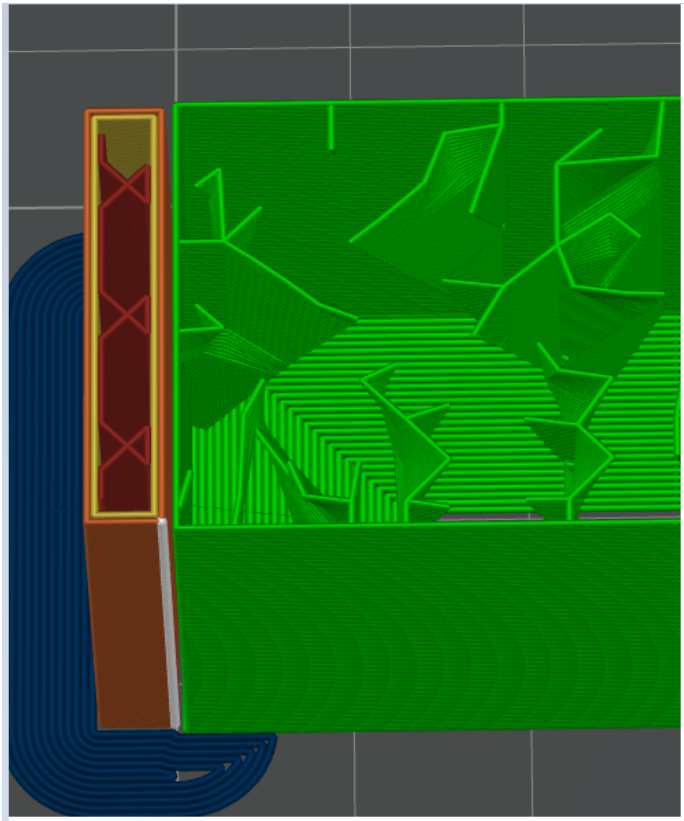
Base pattern: rectilinear



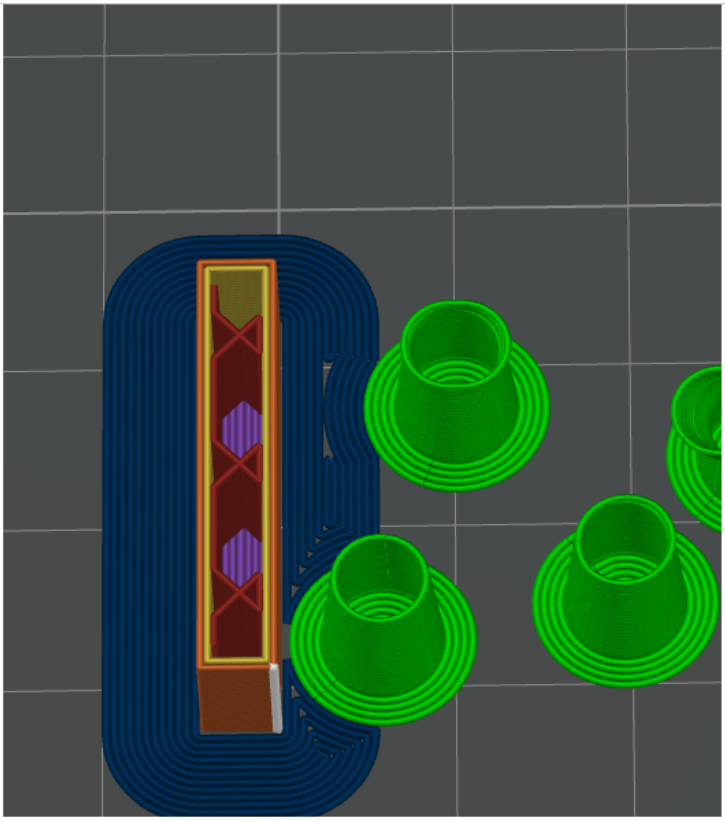
Base pattern: rectilinear grid



Base pattern: honeycomb



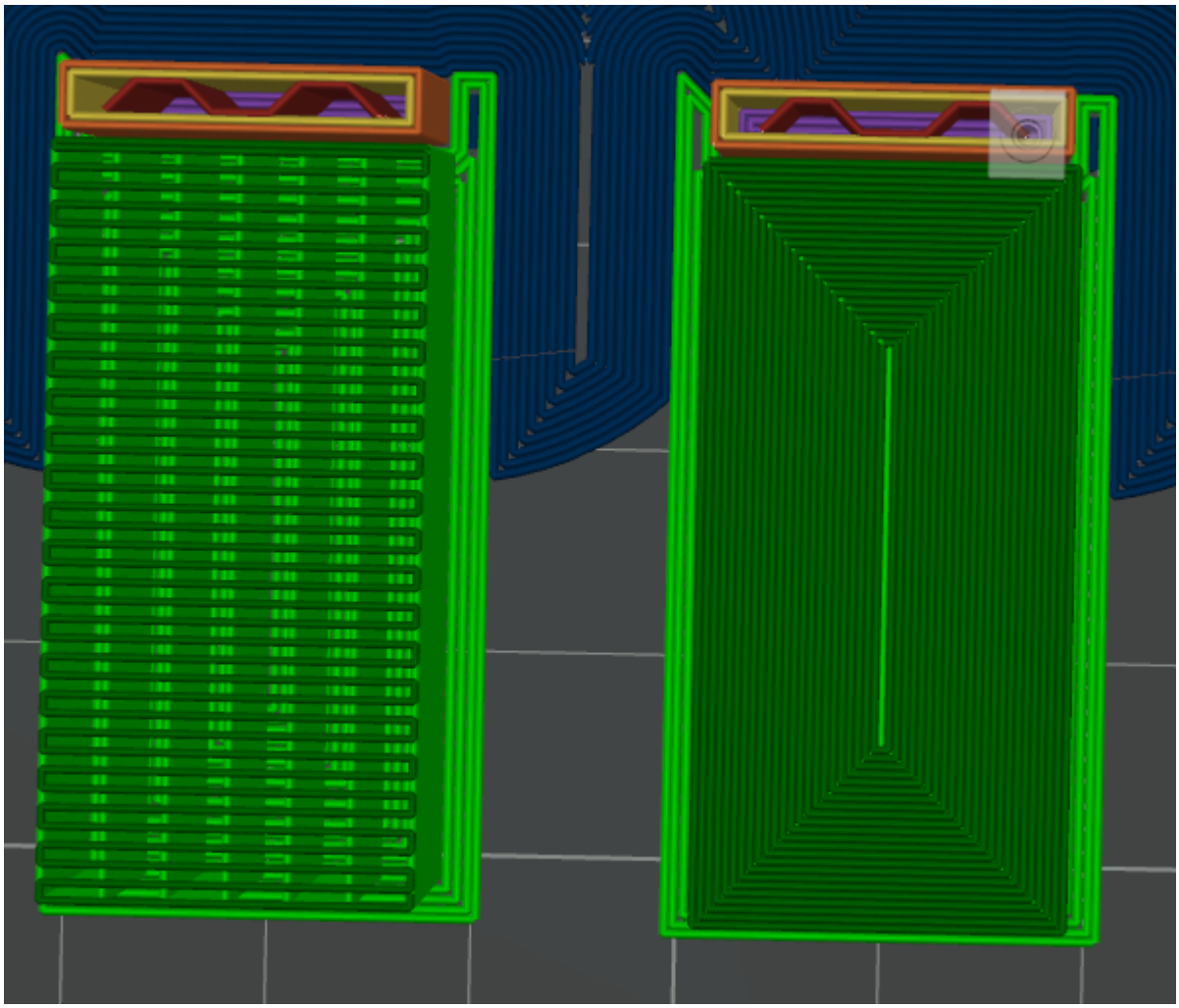
Base pattern: lightning



Base pattern: hollow

- **Base pattern spacing**
  - For **rectilinear** and **rectilinear grid** patterns, this is the spacing between base pattern lines.
  - For the **honeycomb** pattern, this is the radius of each honeycomb cell. So when this value is set to 0, the **honeycomb** pattern degenerates to **rectilinear**.
- **Pattern angle**
  Set the rotation Angle of the support pattern on the horizontal plane.
- **Top interface layers**
  The number of top interface layers. The overhang quality can be improved if we increase this value, at the cost of slightly more material.

- **Interface pattern**
  The line pattern of interface layers. There are currently 3 patterns available:
  - **Rectilinear**: rectilinear pattern, suitable for most cases.
  - **Concentric**: concentric circular pattern, which is stronger on non-planar surfaces and useful with support materials. For best surface quality we can set a very small interface spacing (e.g. 0) when using cocentric pattern and support material.
  - **Default**: kind of auto pattern. The default pattern with support material is rectilinear and concentric with other materials. Support materials may be soluble or not.

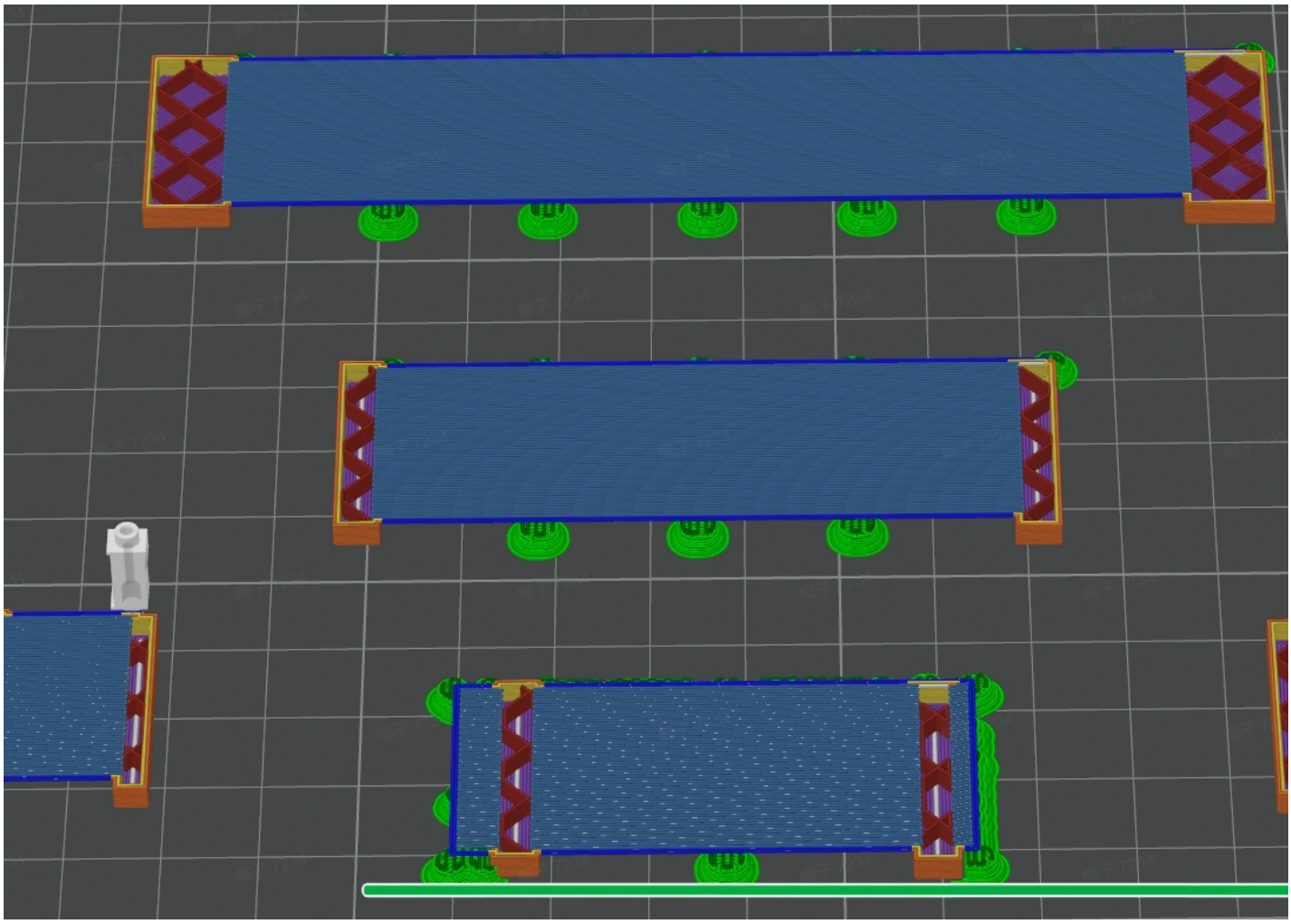


- **Don't Support bridges**
  For normal support, this option controls whether to remove supports for bridges. For tree support, we replace this option with **Max bridge length** which will be explained later.
- **Thick bridges**
  If enabled, bridges will be extruded with higher flow, which means bridges are more reliable and can bridge for longer distances. However, the overhang surface quality may be worse because of possible overflow.

Tree support-only options

Tree support has more options.

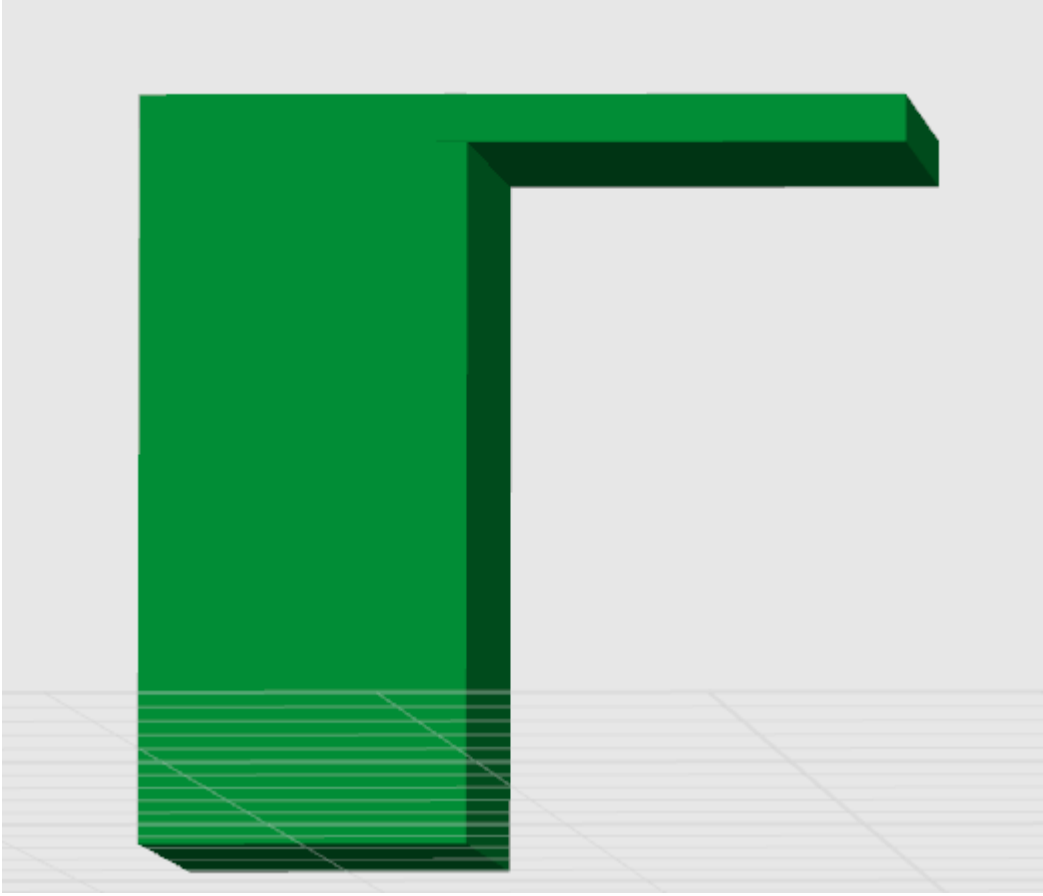
- **Tree support branch distance**: the distance between neighboring tree support nodes. A smaller value means higher sampling density on the overhang surface and, therefore better surface quality, at the cost of more removal difficulty.
- **Tree support branch diameter**: the initial diameter of the tree support node. A larger value means stronger tree supports, also more difficult to remove.
- **Tree support branch angle**: the angle of tree branches stretching out. Larger values mean that tree support branches can be printed more horizontally, with a higher ability to avoid objects and extend further out.
- **Tree support with infill**: infill will be generated in the tree support base if enabled. This makes tree support very strong, so we disable it by default. But if you are using some weak material, e.g. silk PLA, it's suggested to enable this option.
- **Max bridge length**: the max allowed bridging length for overhangs. If an overhang is rectangular, it is regarded as a bridge. A short bridge can be printed well enough without support, because the two ends of the extrusion lines are well supported. The max allowed bridging distance may be different for different materials. When a bridge is larger than max bridge length, it's divided into equal segments and only the contacting points are supported.

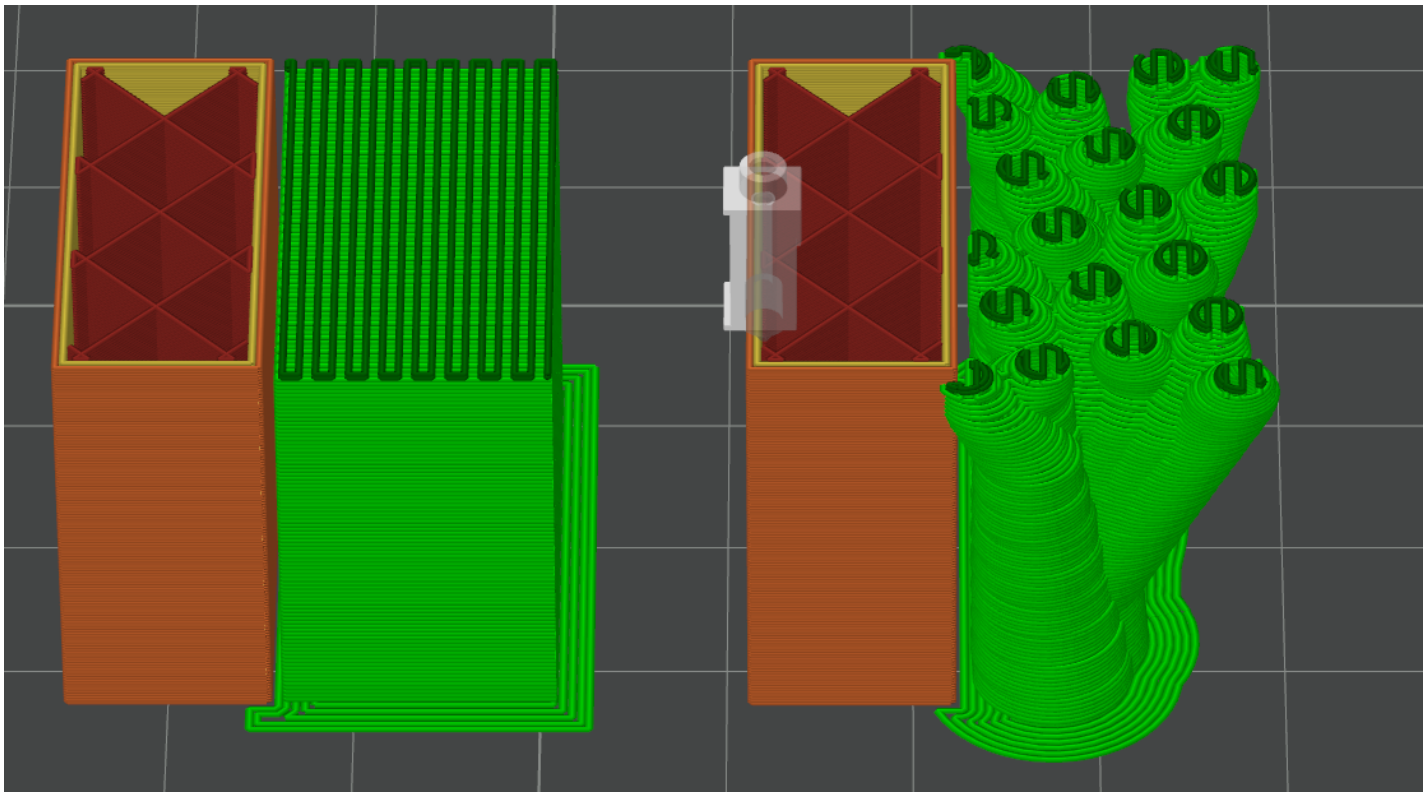


Suitable cases for each type

Normal is better

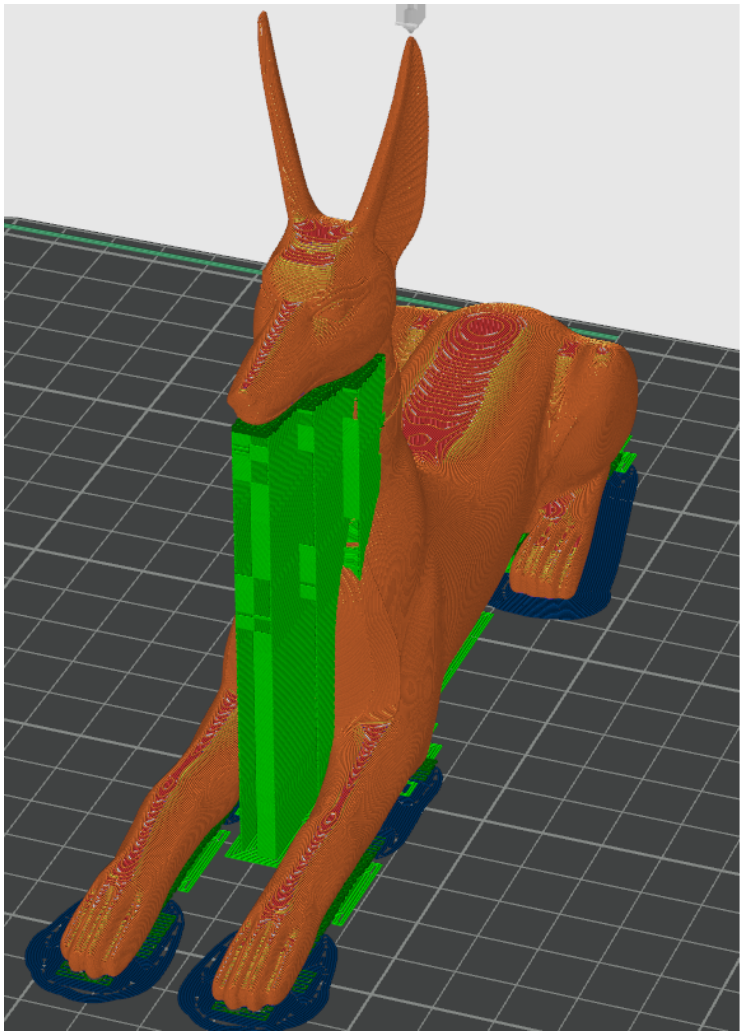
For large planar overhang, Normal supports usually give better surface quality than tree supports. That's why we propose hybrid support. So it's safe to choose hybrid(auto) in general, since for these cases hybrid(auto) will degenrate to normal.

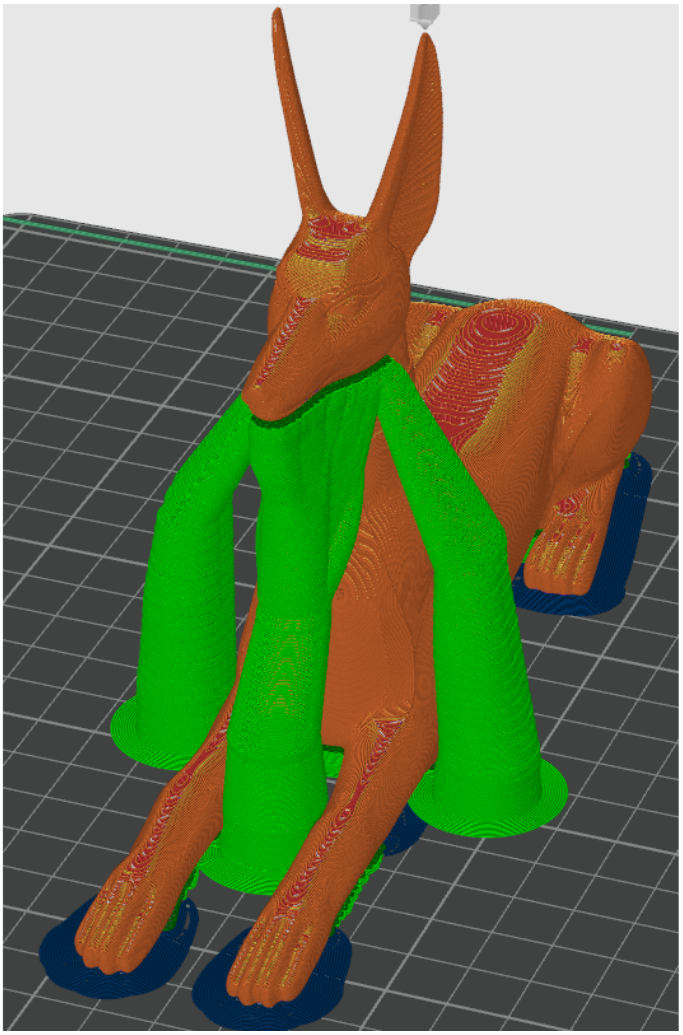




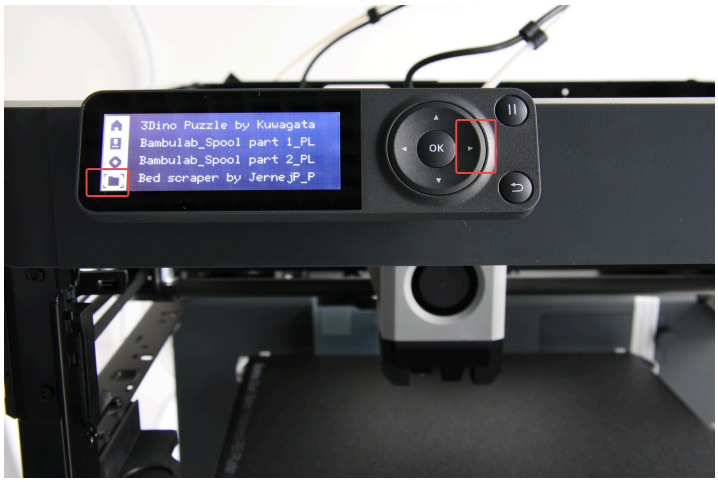
Tree is better

For objects with complex structures and most of the overhangs are small, non-planar surfaces, tree or hybrid(auto) supports give stronger support structure, less material, and less time cost, while keeping similar surface quality.

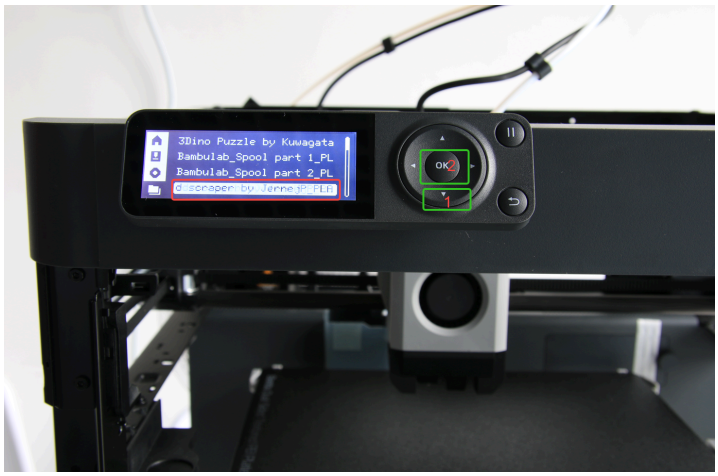




After you've selected the folder icon, use the right arrow to navigate to the pre-load micro SD card file directory.



After entering the pre-load micro SD card file directory, use the up and down arrows to browse the contents of the folder, and then press the OK button to select the model you wish to print.



Confirm the selected model one more, then push the OK button, and the printing will begin.