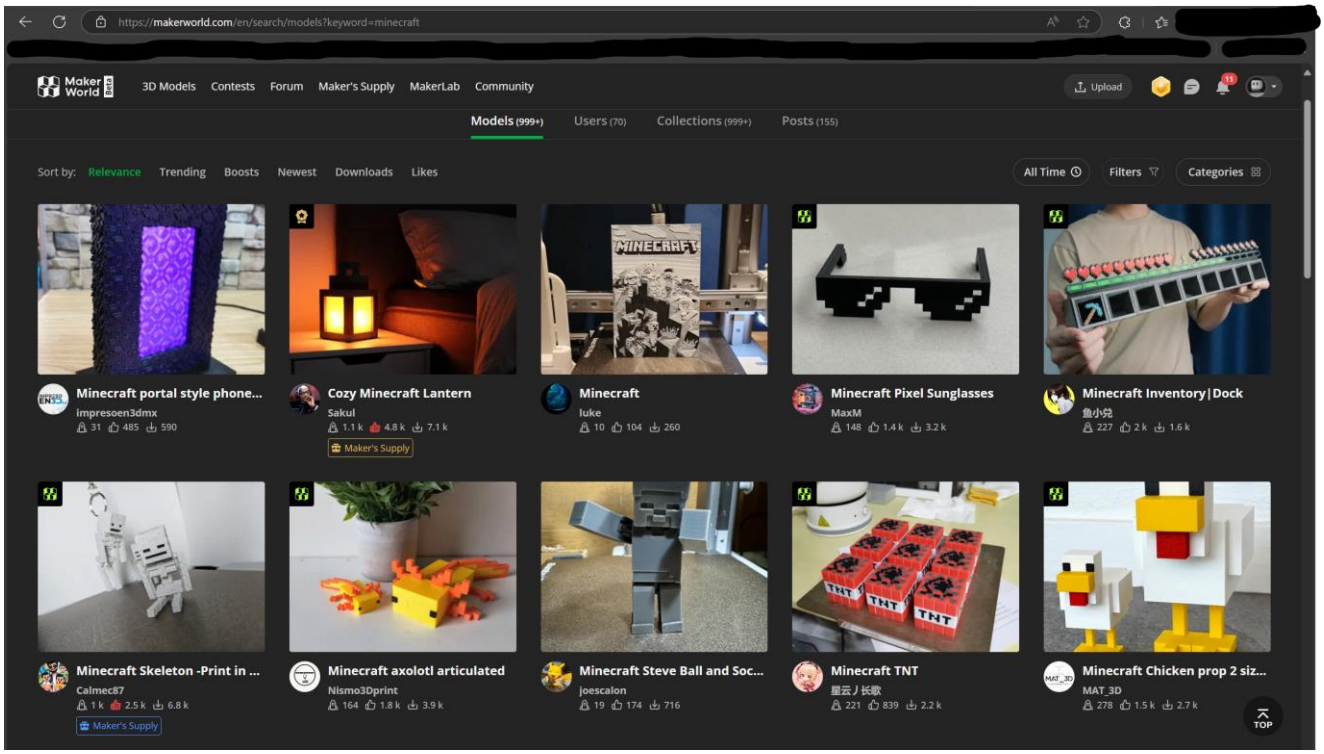
HOW-TO GUIDE

A comprehensive and easy guide on how to order and 3D print with us!

MAKE-IT LABS
See it. Dream it. Make it.

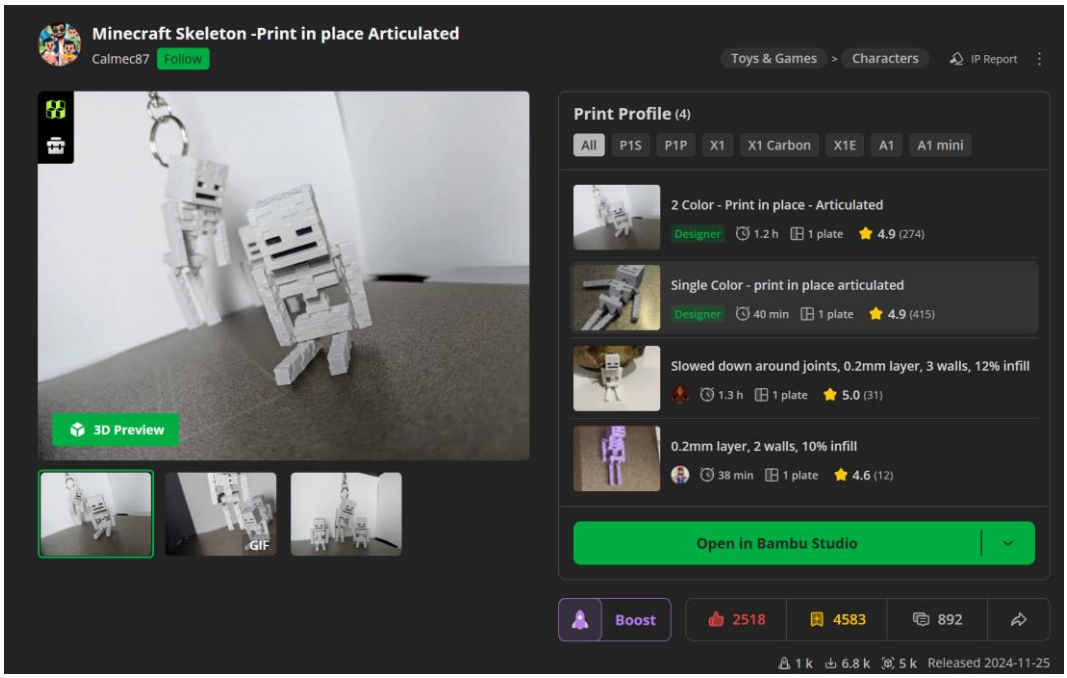
Option 1: Printing 3D models from the online websites (*example: makerworld.com*)

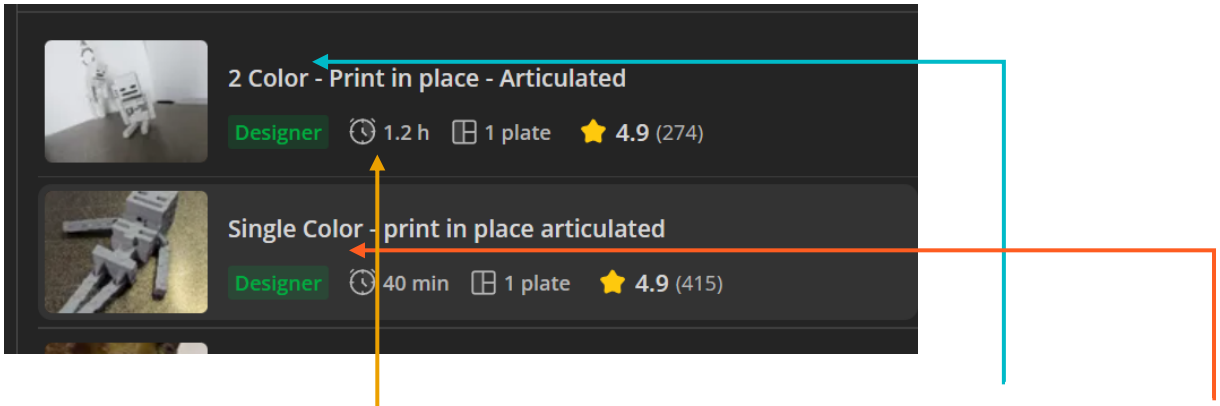
1) Head over to makerworld.com and search for 3D models you're interested to print!



Or alternatively, visit over to Thingiverse (https://www.thingiverse.com) too!

2) In this example, I'm interested to print this Minecraft skeleton model.



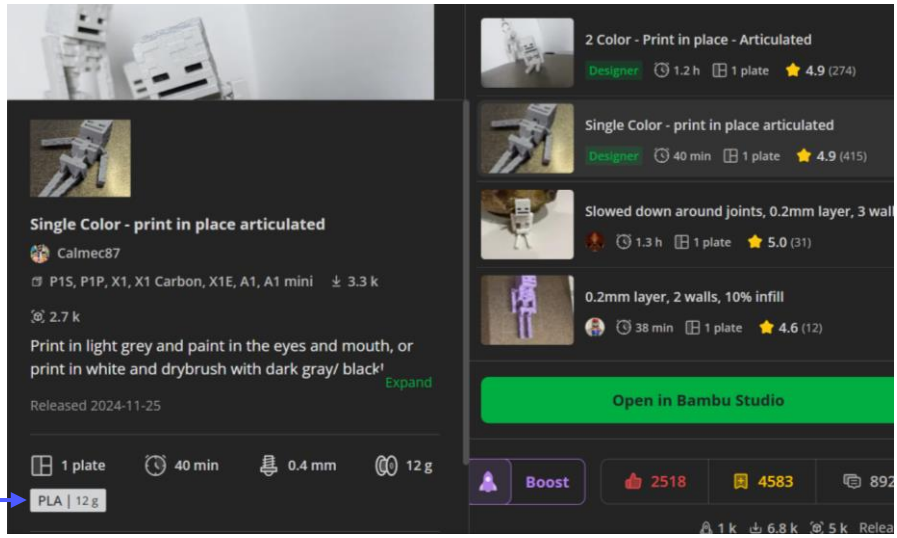


Here, you can see there are 2 options for this model, being multi-colored (2 Color), and Single Color.

1.2 h indicates the time required to print the model. Bigger model requires longer printing time!

3) When choosing the model's printing option, some models offer multi-color, while others only having single color. In this case, the models have both multi-color and single color option:

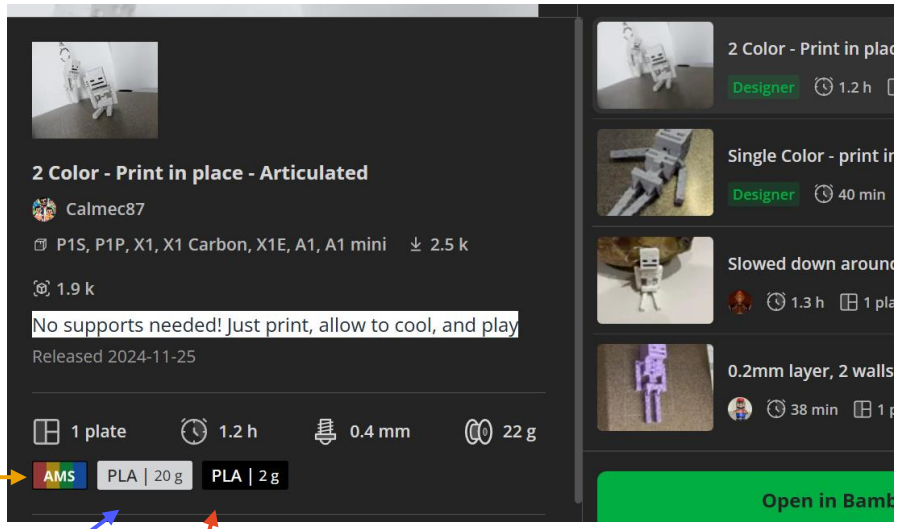
Single color option



Filament usage
(12grams - white)

Multi-color option

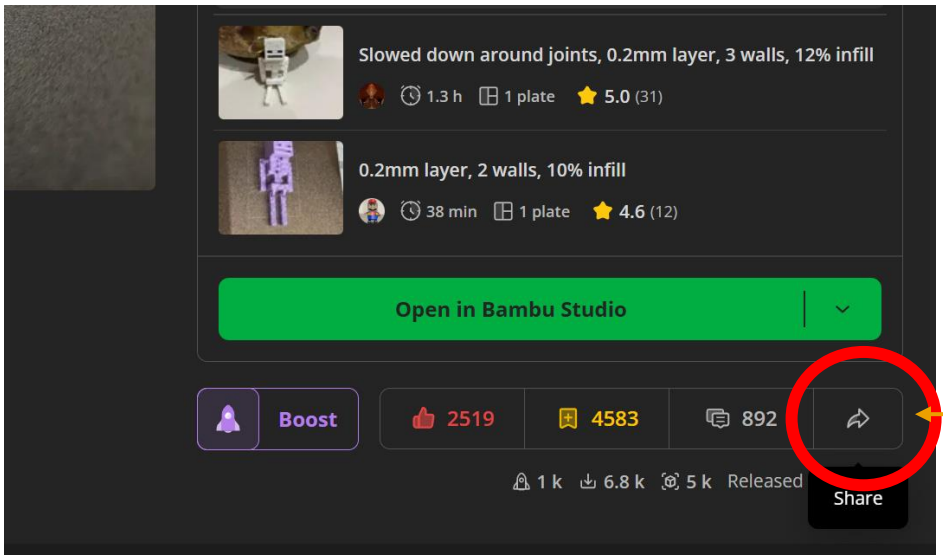
Here, the AMS label means that this 3D Model supports multi-color printing, and also labels which color is used for the printing process

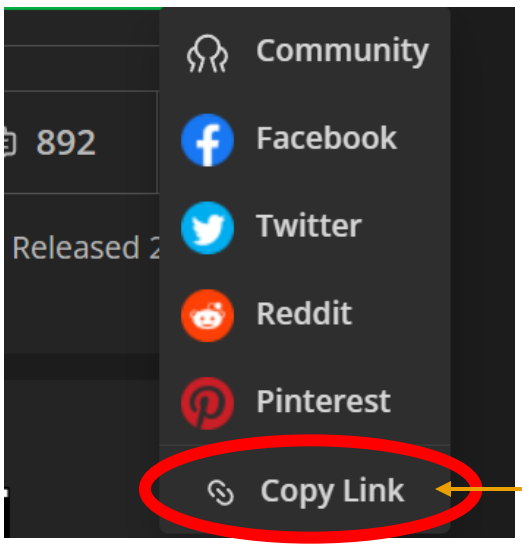


Filament usage
(20grams – White, 2grams - Black)

4) Share the link with us!

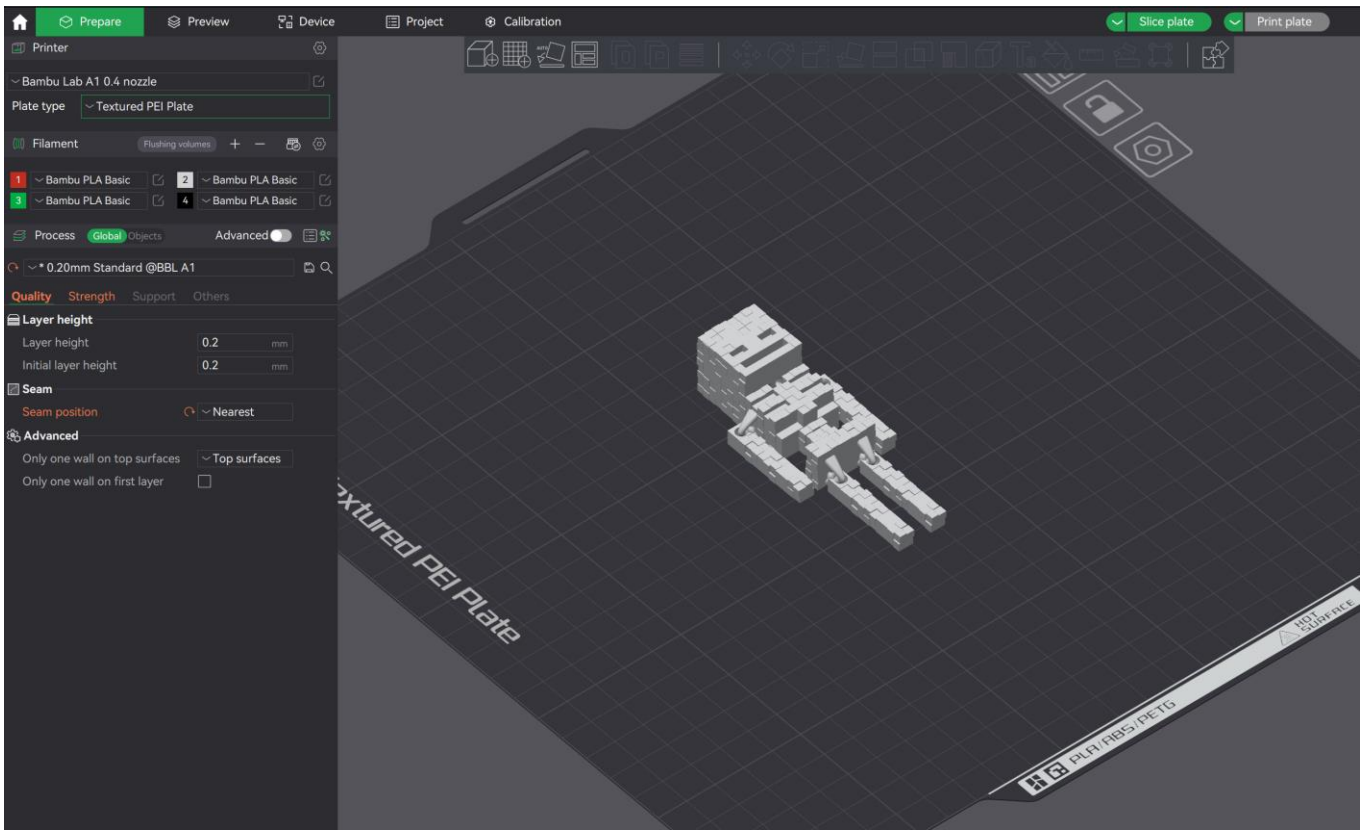
After choosing your desired 3D Model and options to be printed, share the link with us!





The link should look like this:
https://makerworld.com/en/models/820195-minecraft-skeleton-print-in-place-articulated?from=search#profileId-777534

5) After calculating the price based on the weight of the model and payment has received, we will start to begin the 3D printing process of your desired 3D model!



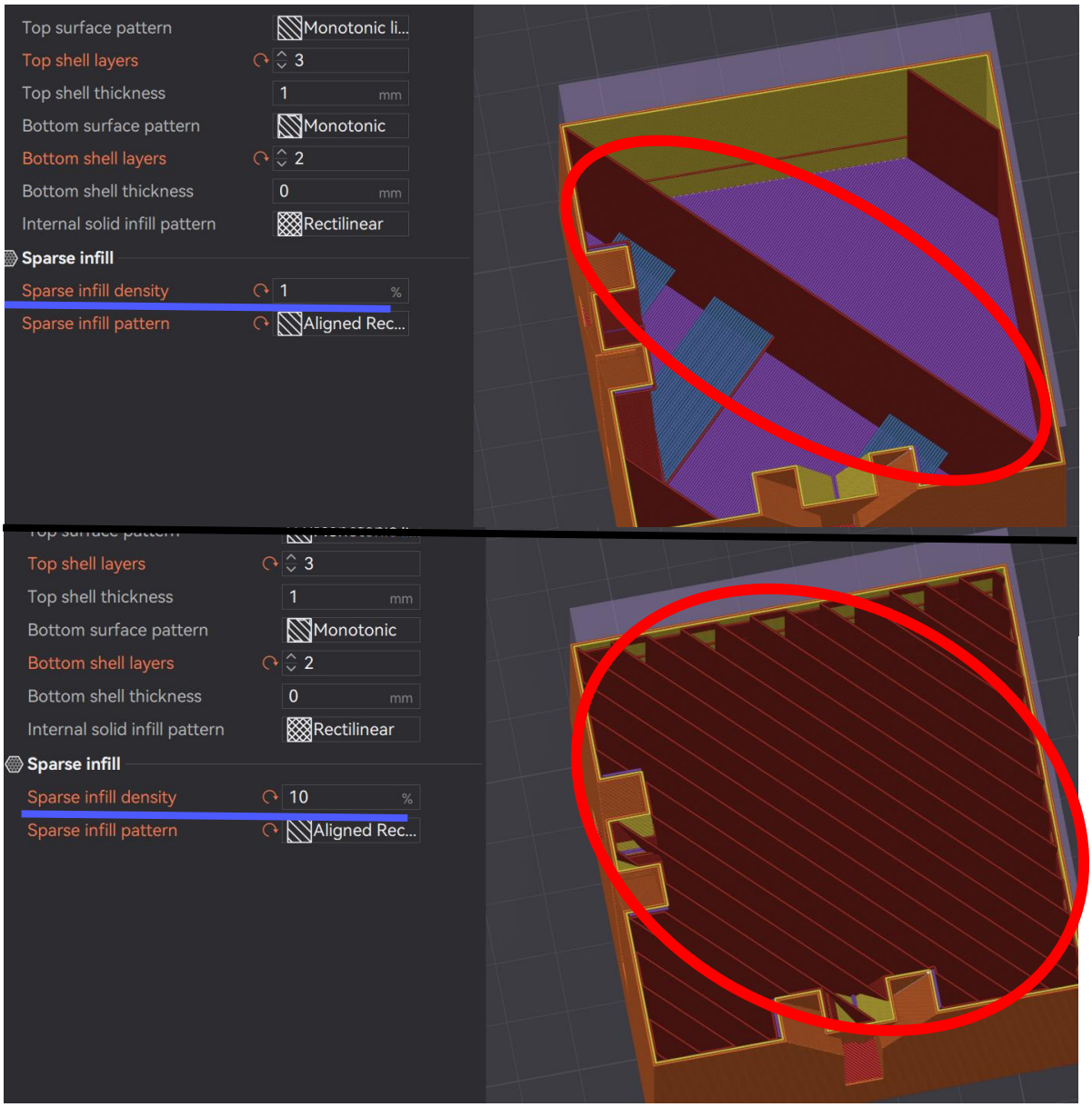
ADVANCE SETTINGS!

For some individuals, they might want to consider to change some few settings on their 3D models. But some of the settings important and worth to note are:

1. SPARSE INFILL DENSITY PERCENTAGE = **How solid/dense or hollow your print is**
2. SPARSE INFILL PATTERN = **Internal structure, example: grid, gyroid or honeycomb**

For individuals who want ease of mind and hassle-free 3D printing and wish not to do complex customization of your 3D models, you are free to sklp this part!

ADVANCE SETTINGS! (skip this if you just want to follow the recommended settings and don't want to customize your 3D model's settings)

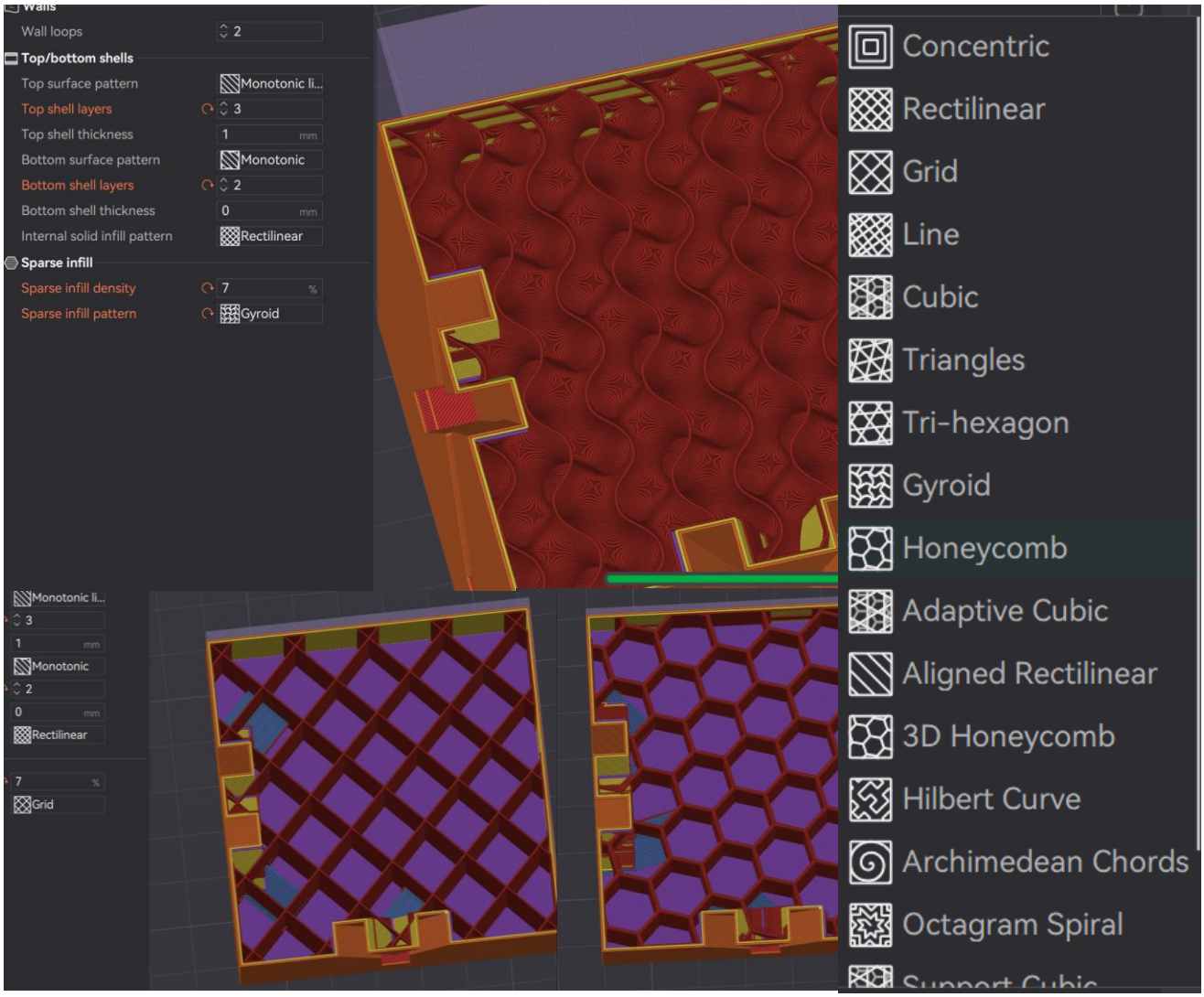


1. SPARSE INFILL DENSITY PERCENTAGE = **How solid/dense or hollow your print will be**

When adjusting this settings, you might be able to save some few grams worth of filament in big size objects, thus resulting at a much cheaper printing price cost. However, lower sparse infill density tends to result in a more "less sturdy" feeling of an object and it will result in much more less impact resistant. For example if a 10cmx10cmx10cm cube with a sparse infill density of 1% were to fall on a ground from the height of a dining table, that cube will dent and is irreversible to fix the damage happened to it.

In general, the recommended setting for sparse infill density would be 7~15%, which will still provide that "rigid" and "tough" feeling, while still being budget friendly.

ADVANCE SETTINGS! (skip this if you just want to follow the recommended settings and don't want to customize your 3D model's settings)



2. SPARSE INFILL PATTERN = **Affects the internal structure, example: grid, gyroid or honeycomb**

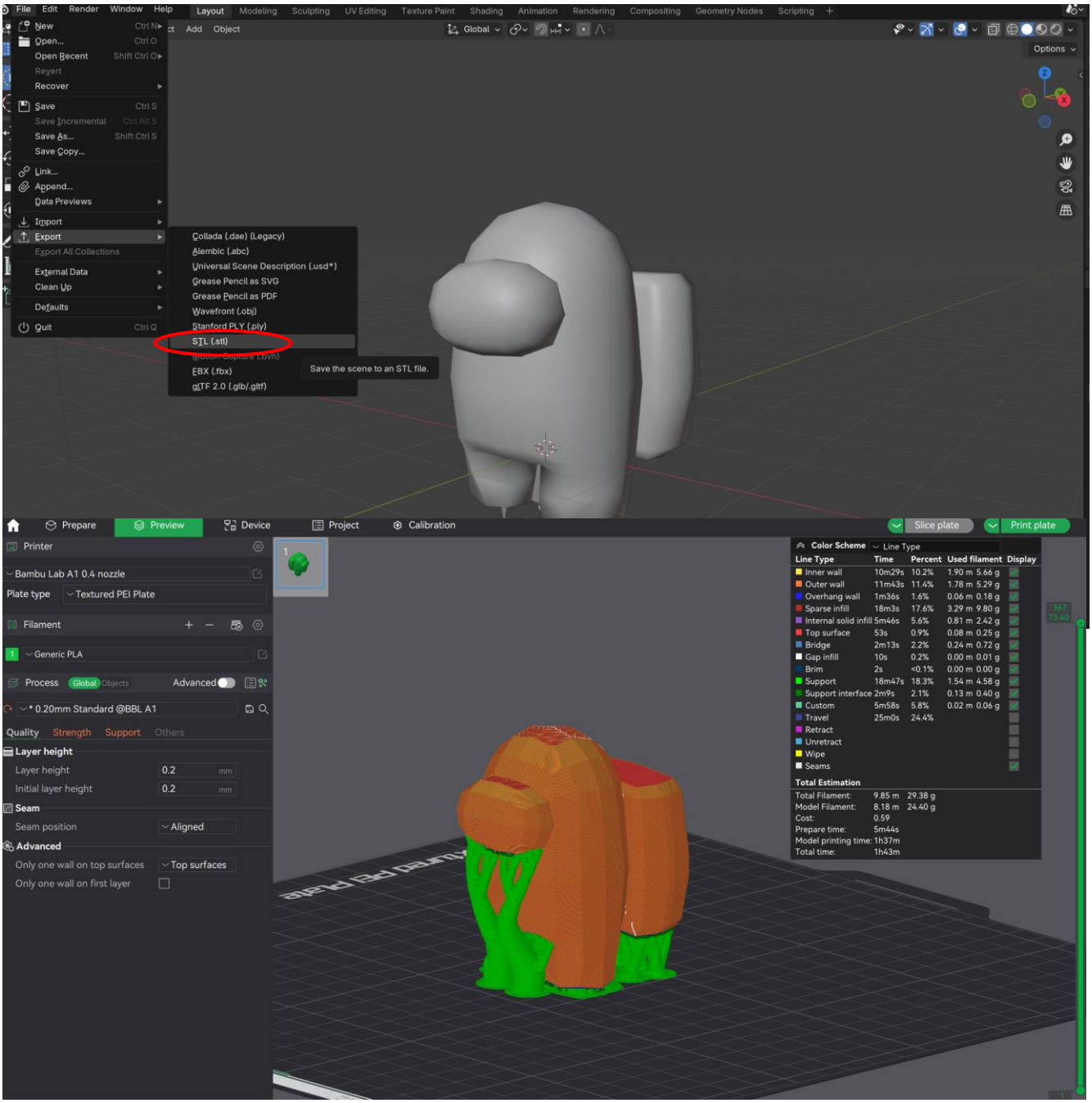
Adjusting this setting will affect the strength properties of your object!

In general, the recommended setting for sparse infill pattern would be **gyroid**, which will provide the good strength, rigidity and impact resistance of an object, and suitable for every 3D model while still being efficient and won't consume too much filament (i.e, not expensive to print). There is a list in the picture above, listing out the available patterns to choose from to be made as the infill pattern

---

There are also more advance settings that can be adjusted on the 3D printer such as layer height, surface pattern, fuzzy skin, and etc. However certain settings do increase the usage of certain components, such as an increase usage of electricity consumption, thus certain adjustment would need to cost an additional small fee (RM1~RM10). If an individual do want to do further setting adjustments, just let the team know and we are free to help and guide you!

Option 2: Printing personal hand made 3D models (i.e, Blender, Fusion 360, SolidWorks, etc.)



1) Export your personal 3D object file extension to a .STL or .3mf file type (or into any Bambu Studio supported file type [STEP, SVG, AMF, and OBJ])
2) Share the file to MAKE-IT LABS by contacting us
3) We'll process and print your 3D model!

**The green things are called tree support. It is generated when there are 'overhangs' of which the 3D printer can't reach to print those parts and thus needing those support to print the 'levitating' areas. Don't worry as they are easily removable after the printing process won't effect the quality.*