CreatBot
D600
Professional
Large Volume
3D Printer

CrEatBot D600 Professional Large Volume 3D Printer User Manual

Contents

CreatBot

CrEatBot D600 Professional Large Volume 3D Printer

Specifications

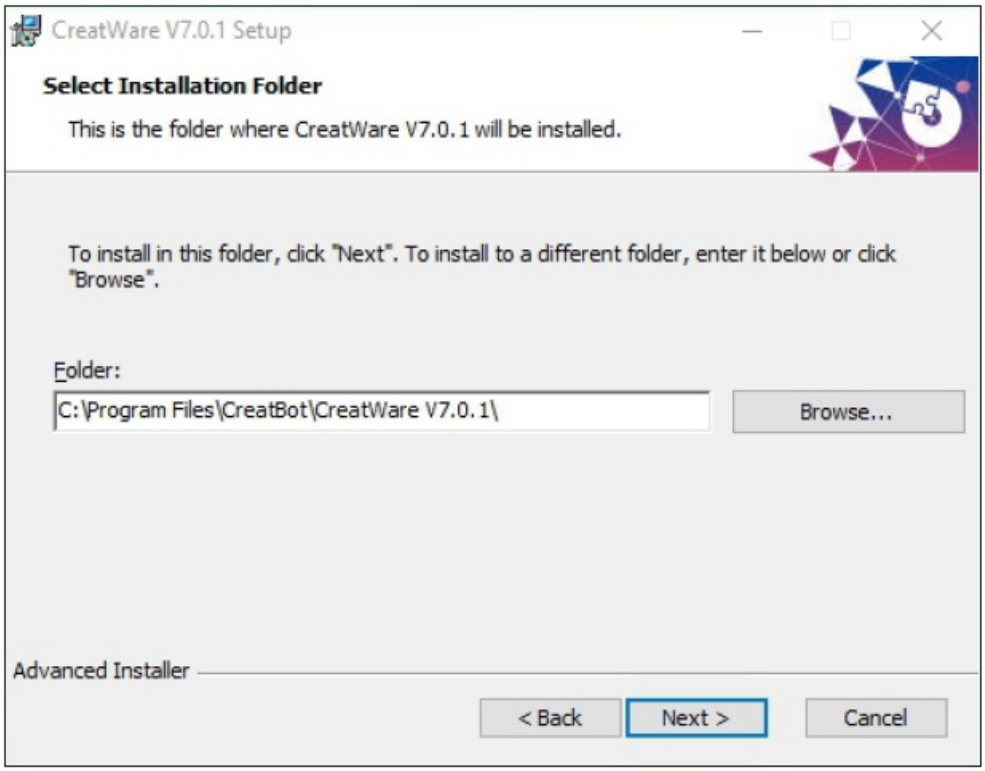
- **Product Name:** CreatWare V7.00
- **Manufacturer:** Henan Creatbot Technology Limited
- **Website:** www.creatbot.com.

FAQs

- **Q: How do I import models into CreatWare?**
  - **A:** Click the Add (Ctrl+I) button to import models in STL, obj, or of formats.
- **Q: Can I customize print settings in Expert Mode?**
  - **A:** Yes, in Expert Mode, you can customize printer parameters, Gcode, etc.
- **Q: How do I align a model with the print bed?**
  - **A:** Click Place on Face (F) to align the model with the print bed by selecting white planes.

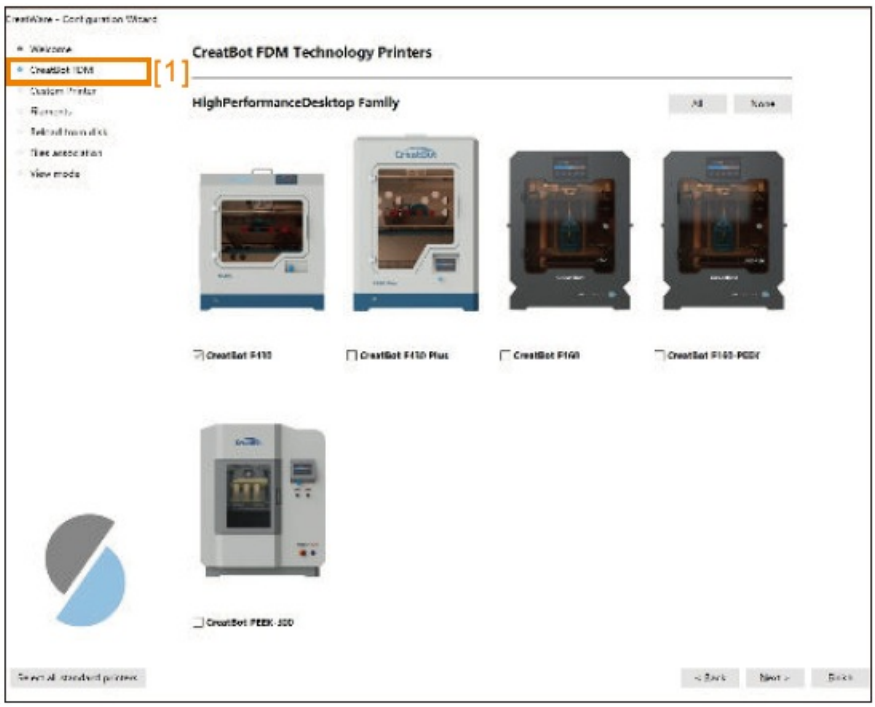
CreatWare Installation

Download the CreatWare installation package from www.creatbot.com, follow the prompts to install the software, and try using the default installation path as well.

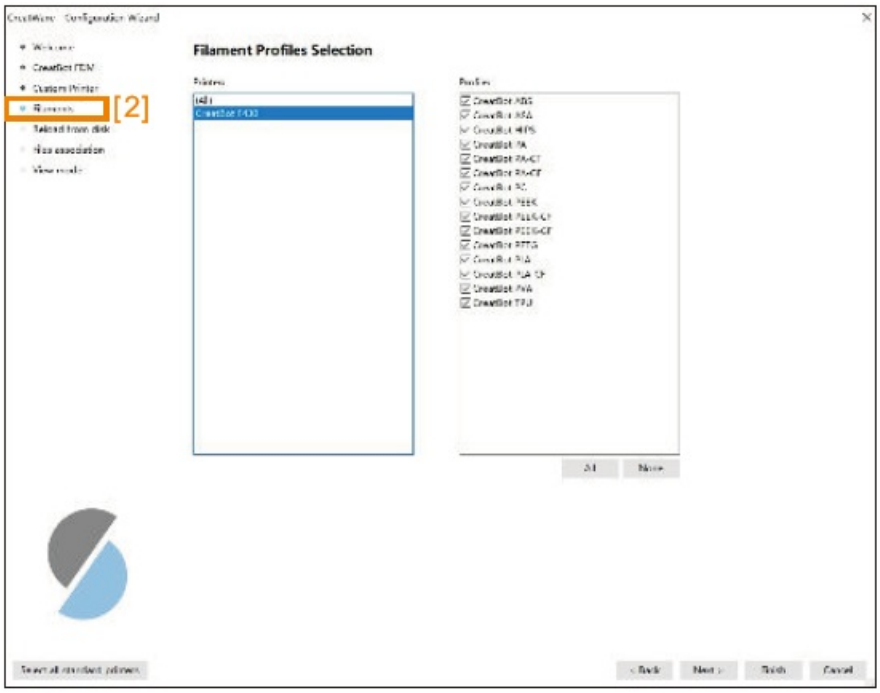


Configuration Wizard

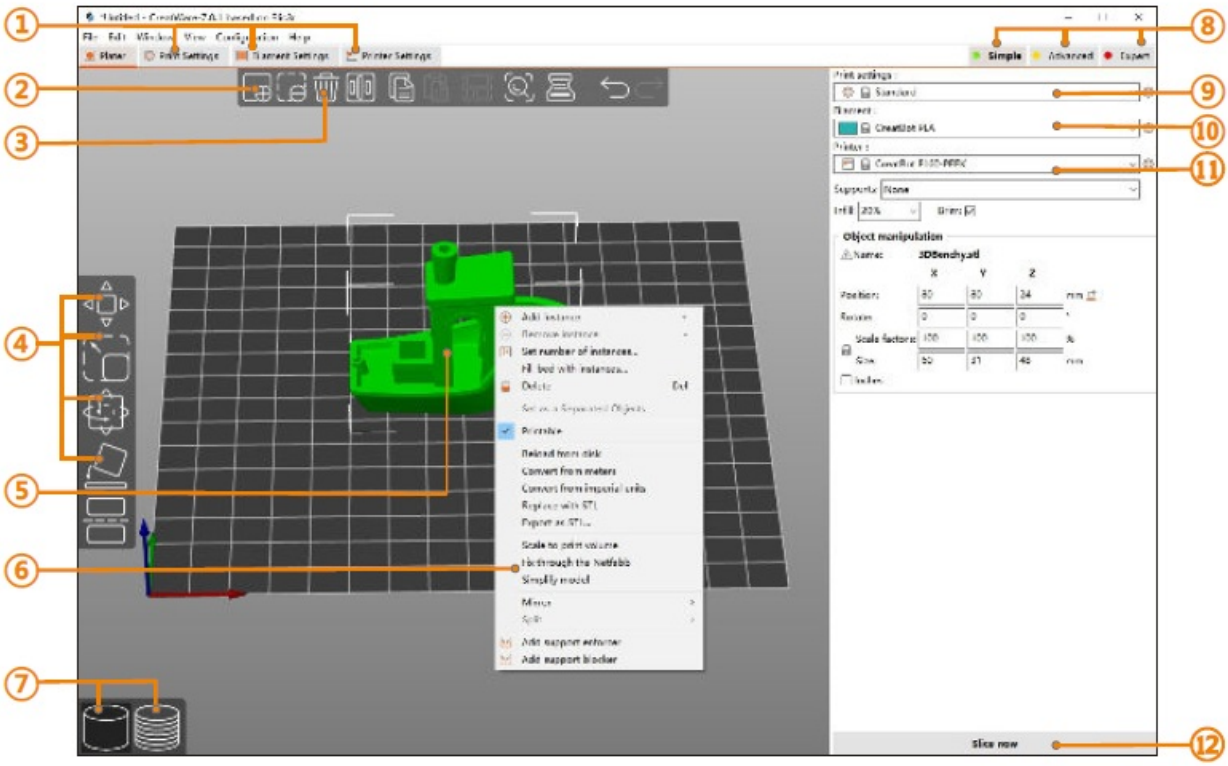
- **CreatBot FDM [1]:** Select your p [1] printer model from the list.



- **Filaments [2]:** Select filament from the list. If need to print other filaments not listed, such as PVDF, Ultem, etc. Select a similar filament first, then change settings in the software, and save.



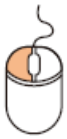
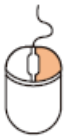
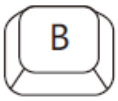
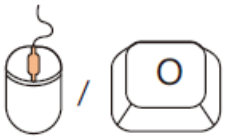
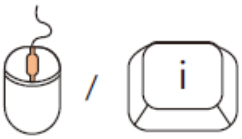
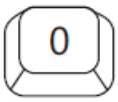
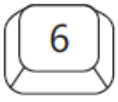
User Interface



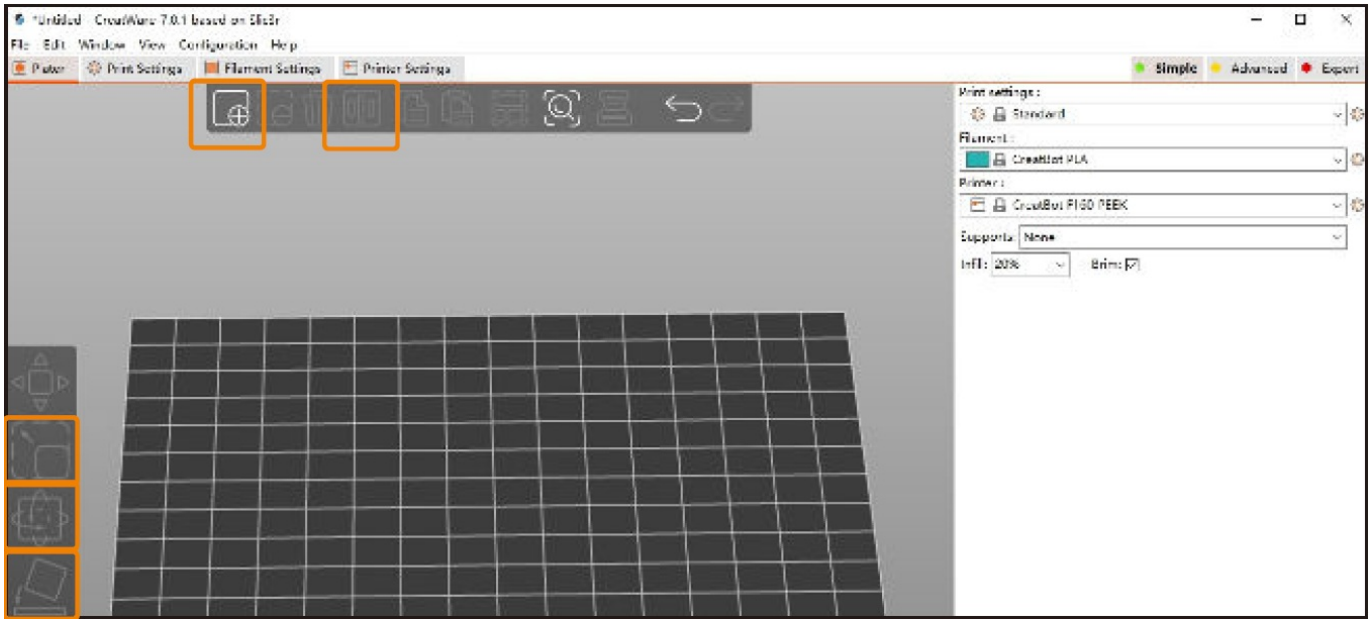
1. Opens detailed settings of print, filament, and printer
2. Add models into CreatWare
3. Delete models from CreatWare
4. Move, Scale, Rotate, Place on plate
5. Model preview
6. Right-click on the model opens a context menu
7. Switch between the 3D editor and layers preview
8. Switch between Simple / Advanced / Expert mode

9. Profile selection
10. Filament selection
11. Printer selection
12. Slice and generate G-code

View

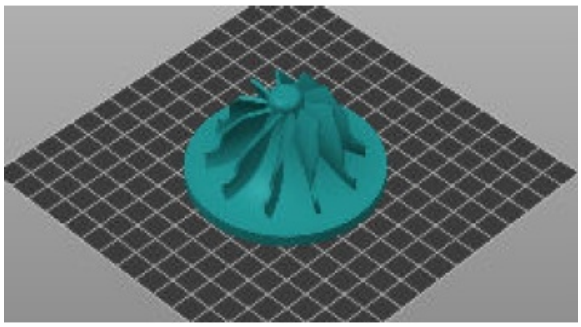
	Rotate		Pan
	Zoom on the entire print bed		Zoom on selected objects or on all objects if none selected
	Zoom out		Zoom in
	Isometric view		Top-down view
	Bottom-up view		Front view
	Back view		Left view
	Right view		

Import Model



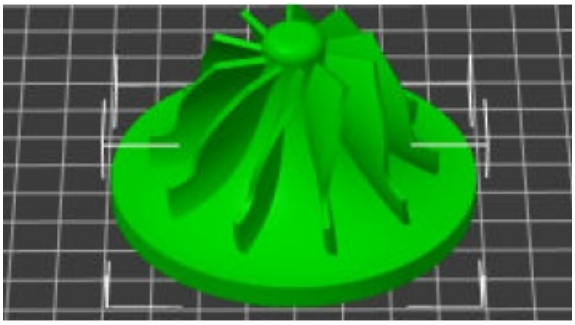
Import model

-  Click the “Add” (Ctrl+I) button to import the models. (STL, obj, and files)



Arrange

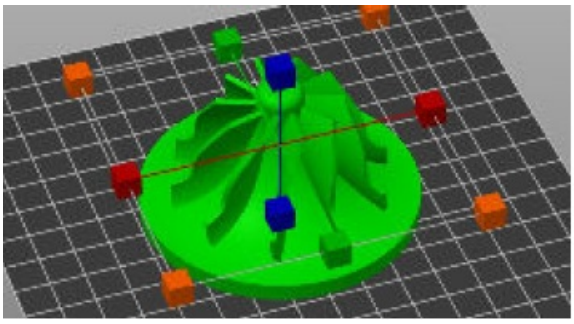
-  Click the “Arrange” (Shift+A) button to auto-arrange the models.



Scale

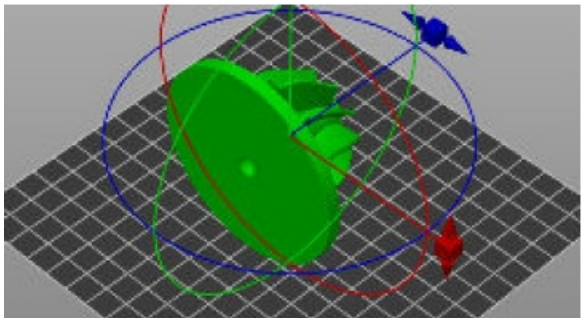
-  Click “Zoom” (S), you can see a contour line around the model. Drag the outline to adjust the size of

the current model. You can input the value of the zoom ratio in Scale X / Y/ Z, or directly enter the size you need.



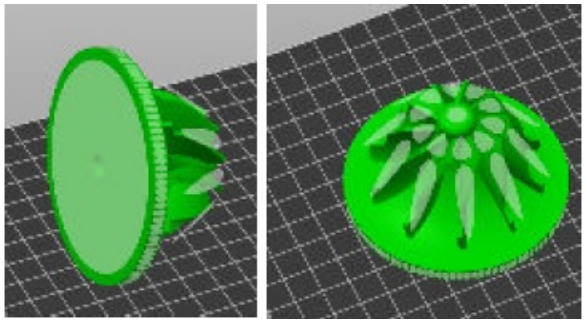
Rotate

-  Click ” Rotate ” (R), and you can see three ting lines around the model, respectively along the X, Y, and Z directions. Drag the pointer to rotate the desired angle. You can also enter a value in the lower right corner.



Place on face

-  Click” Place on the face(F)“ and multiple white planes will appear on the model. Clicking on any of the white planes will align that plane with the print bed.



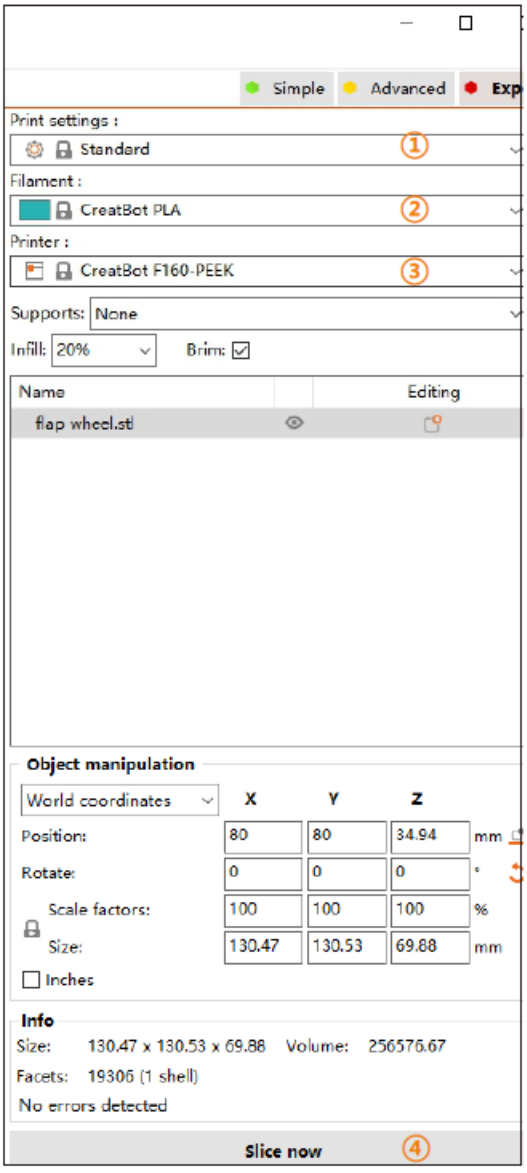
Slicing Operation

Parameter Modes

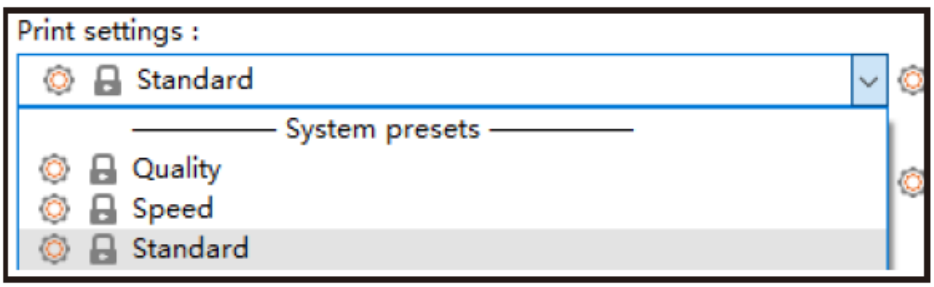
-  **Simple mode**
  - **Simple mode:** only displays the basic parameters, suitable for beginners to print common material.

- **Advanced mode**
  - **Advanced mode** display more parameters, suitable for advanced users.
- **Expert Mode**
  - **Expert Mode:** display all settings parameters, you can customize the printer parameters number, G-code, etc.

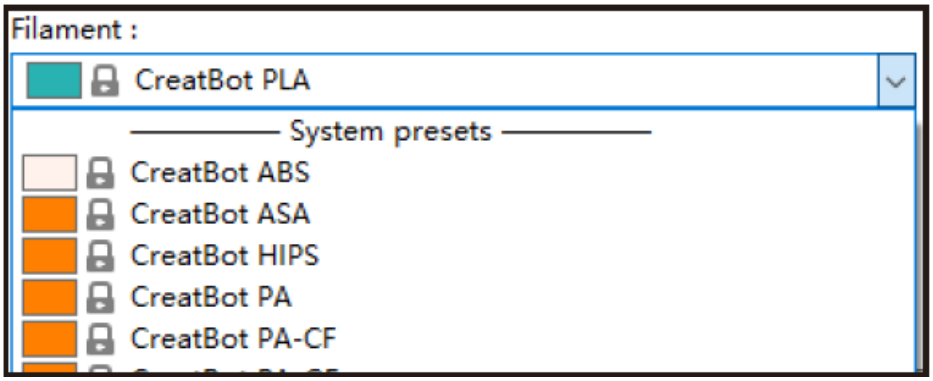
Slicing Step



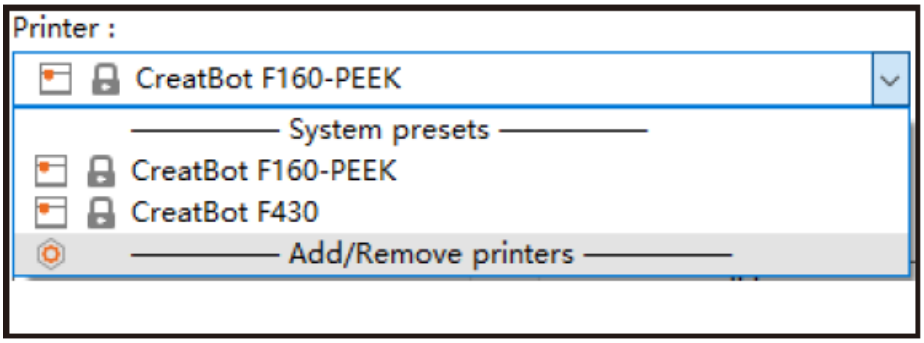
- **Step 1:** Select print settings. Three print settings are provided by default, Standard, Quality, and Speed. You can also change each setting according to your needs.



- **Step 2:** Select filament.



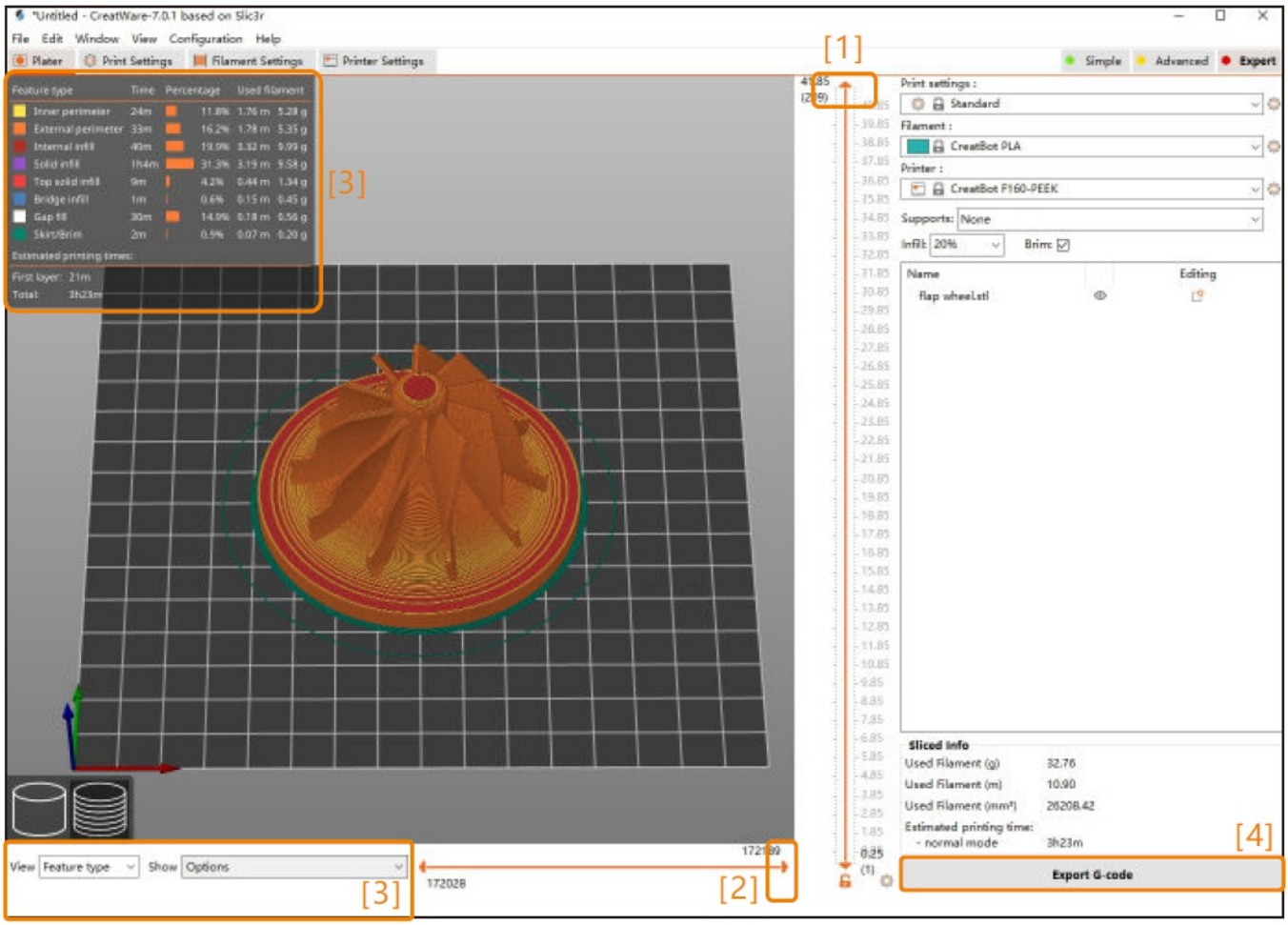
- **Step 3:** Select printer.



- **Step 4:** Click “slice now”

Preview / Export

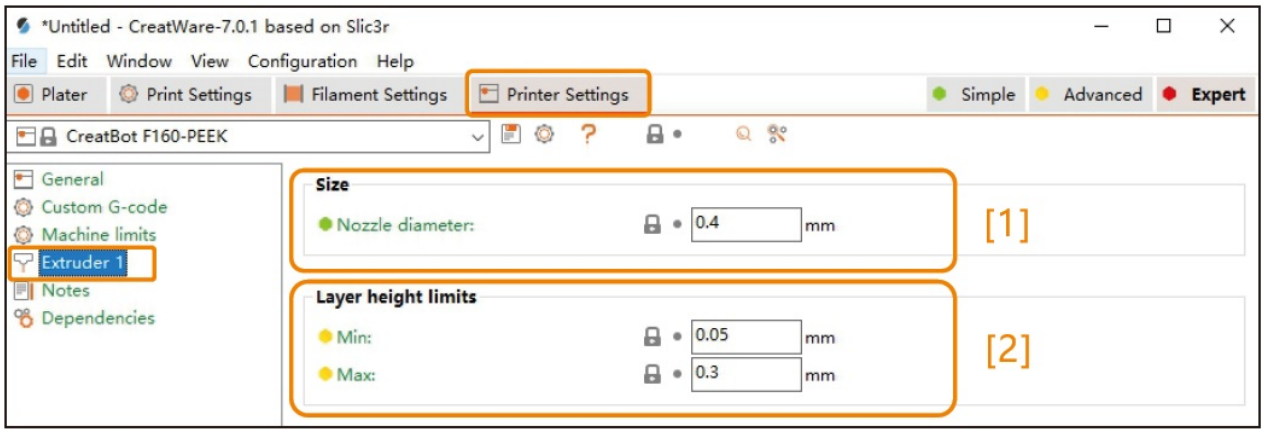
- Preview G-code file: Drag the right indicator bar [1] to preview the layer height and printing status of each layer; drag the lower indicator bar [2] to view the specific information of each layer in detail.
- Each feature can be viewed in the upper left corner [3]. You can also view other printing parameter information, such as temperature, speed, layer height, line width, etc.



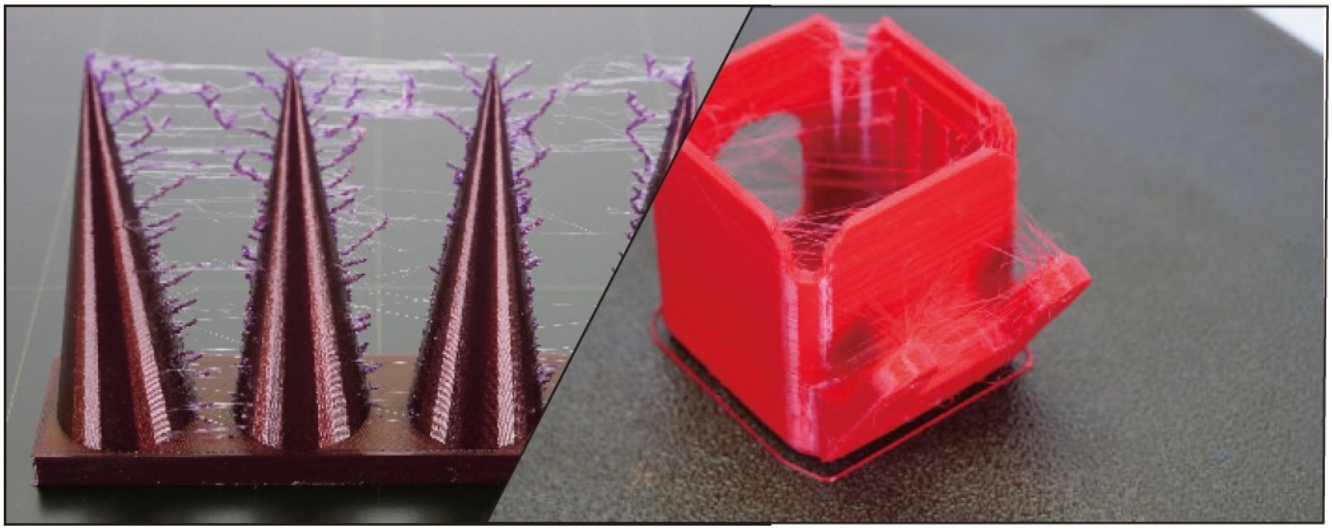
Export G-code: Click Export G-code[4], save the G-code to the U disk, insert the U disk into the printer port, select the file, and start printing.

Common Settings

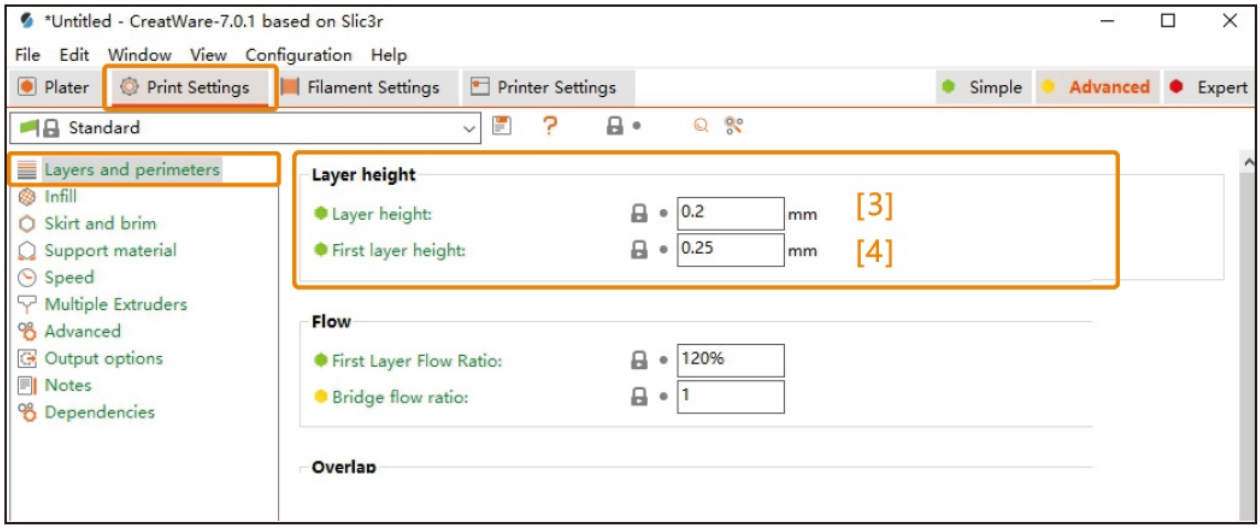
- **Extruder**
  - **Nozzle diameter [1]:** The default size is 0.4mm. [1] If you use a large size nozzle, the extrusion of the filament will multiply and need to increase the printing temperature.



  - **Retraction [2]:** For direct drive extruders (such as F430, PEEK-300, D600Pro), the default retraction distance is 2mm (within 3mm); for Bowden extruders (such as DX, DE), the default retraction distance is 4 -5mm. If the retraction distance is too small and will leave some strings, you can increase the retraction distance. If the retraction distance is too large, the printing time will be affected.
- **Strings picture:**



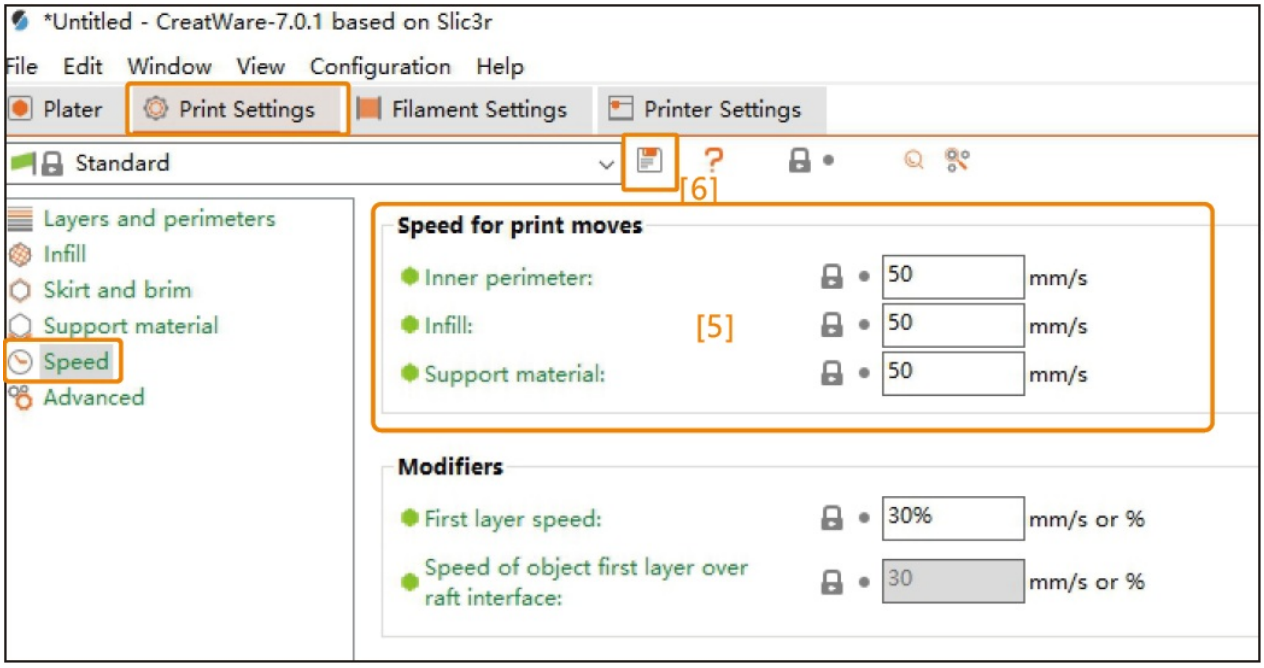
- **Layer**
  - **Layer height [3]:** The most important parameter [3] that determines print quality. Generally set to half of the nozzle diameter, taking into account the printing speed and quality. When the nozzle diameter is 0.4mm, it is recommended to use 0.1mm for high precision and 0.3mm for high speed.



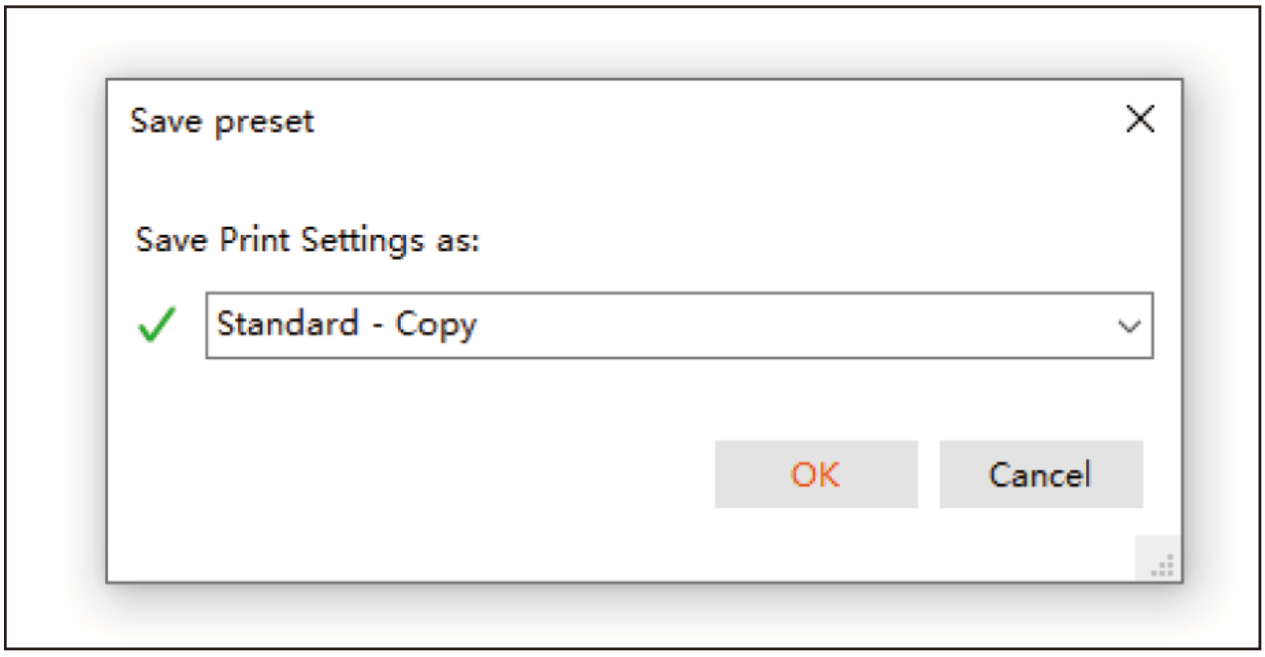
  - **First layer height [4]:** It refers to the height of the first layer of printing, generally greater than 0.2mm. This parameter can compensate for the errors of the platform and make the model stick to the platform better. First Layer Flow Ratio: the extrusion amount of the first layer. A slightly larger extrusion will make the model stick more firmly to the platform.
  - **Layer height range**

Nozzle diameter (mm)	Layer height range(mm)
0.2	0.05-0.15
0.3	0.10-0.20
0.4	0.10-0.30
0.5	0.15-0.35
0.6	0.20-0.40
0.8	0.25-0.5
1.0	0.30-0.60

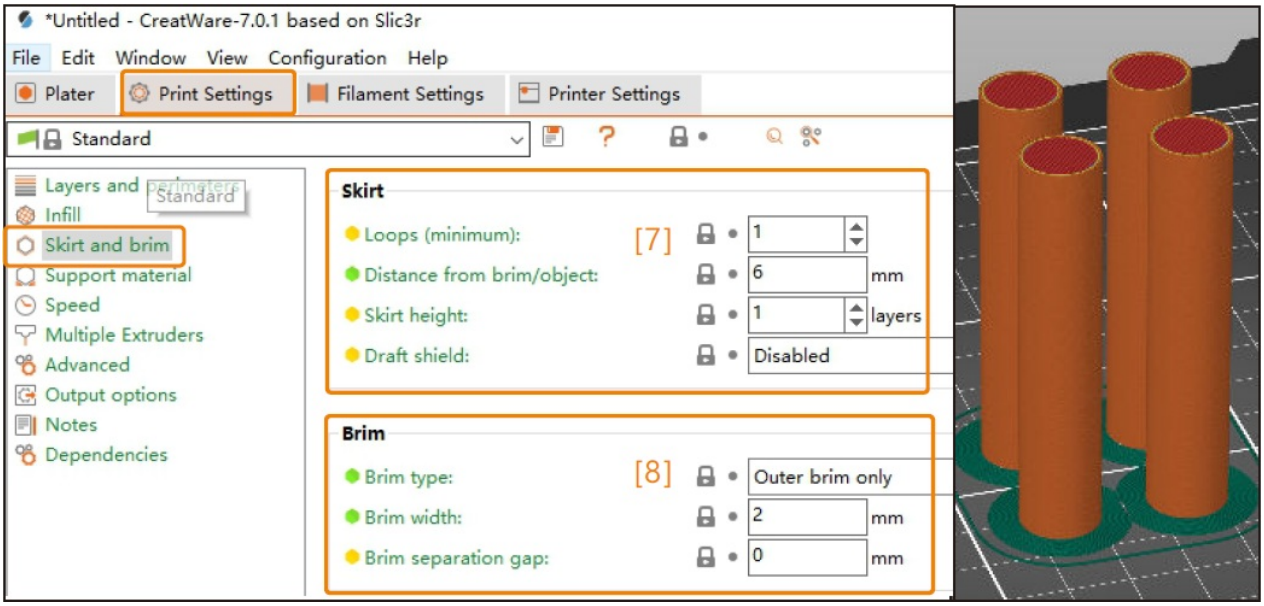
- **Speed**
  - **Speed [5]:** Speed affects the surface quality of the [5] model. The faster the speed, the lower the print quality. The recommended printing speed is 40- 60mm/s. If the printing speed is increased, the temperature of the nozzle needs to be increased to ensure that the filament can be extruded smoothly.



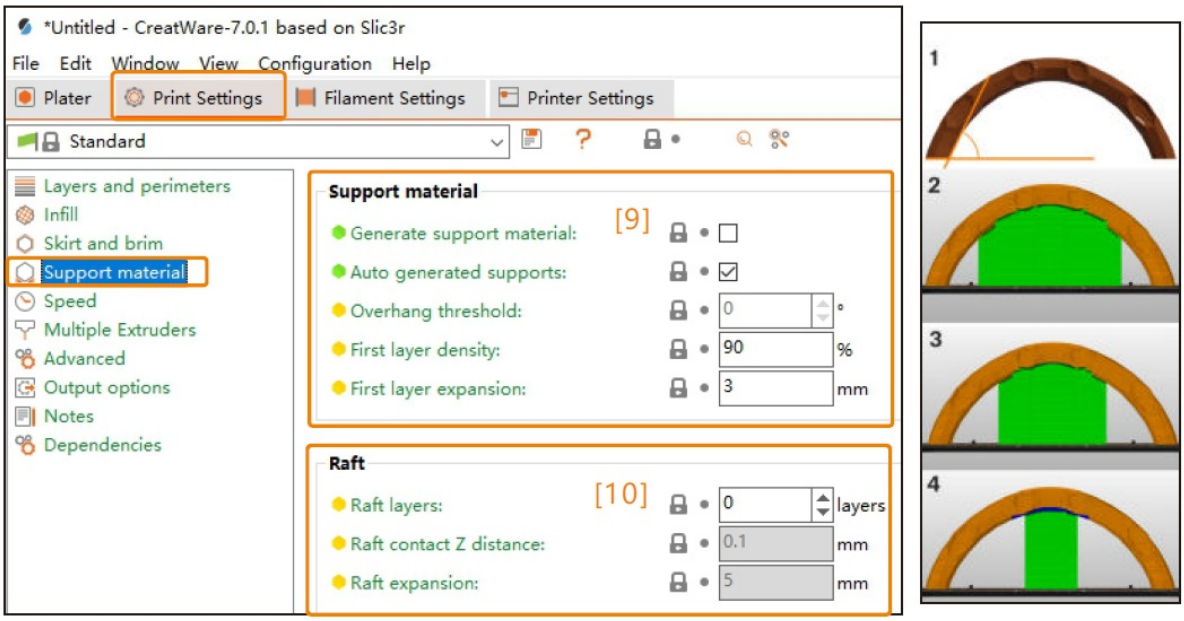
  - **Change printing settings [6]:** After changing the printing parameters, click save and customize the name for future use.



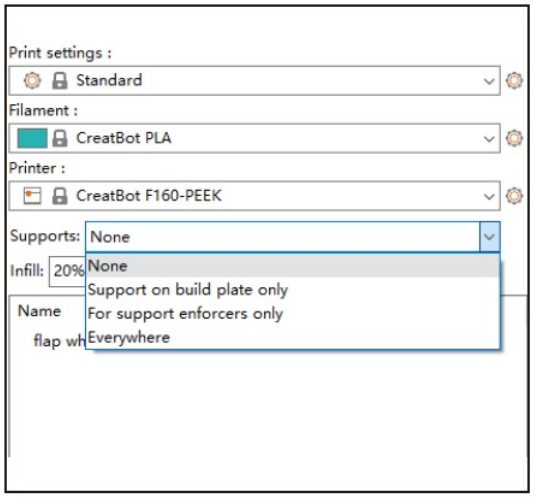
- **Skirt and Brim**
  - **Skirt [7]:** It is mainly used to confirm t [7] the printing area and check whether the nozzle is extruded normally.
  - **Brim [8]:** Print a series of concentric lines around the model to increase adhesion between the model and the platform.

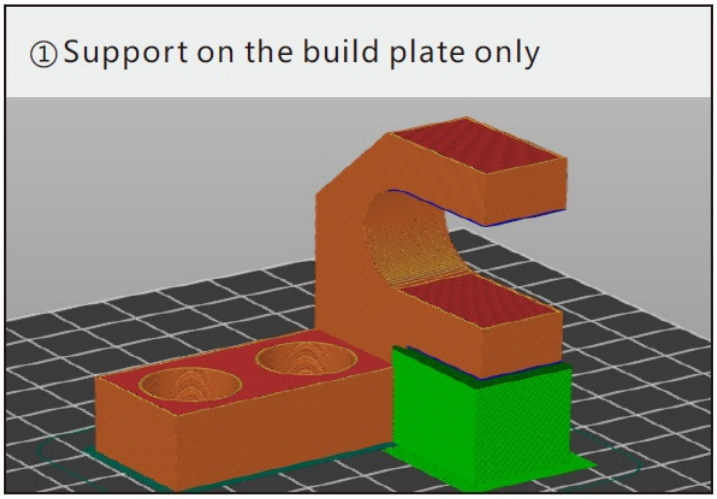


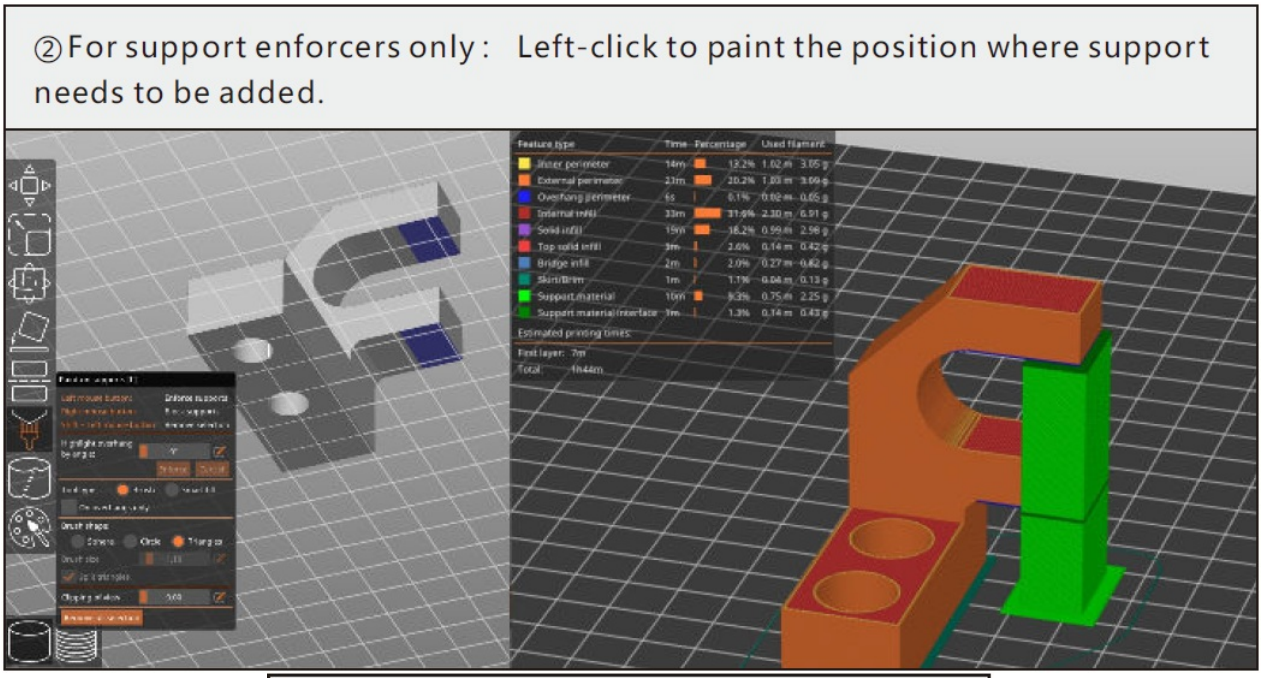
- **Support**
  - **Support [9]:** Add support to the overhanging area of the model.
    - **Note:** The smaller the angle of support, the less support. Set to 0 to automatically detect and add support.

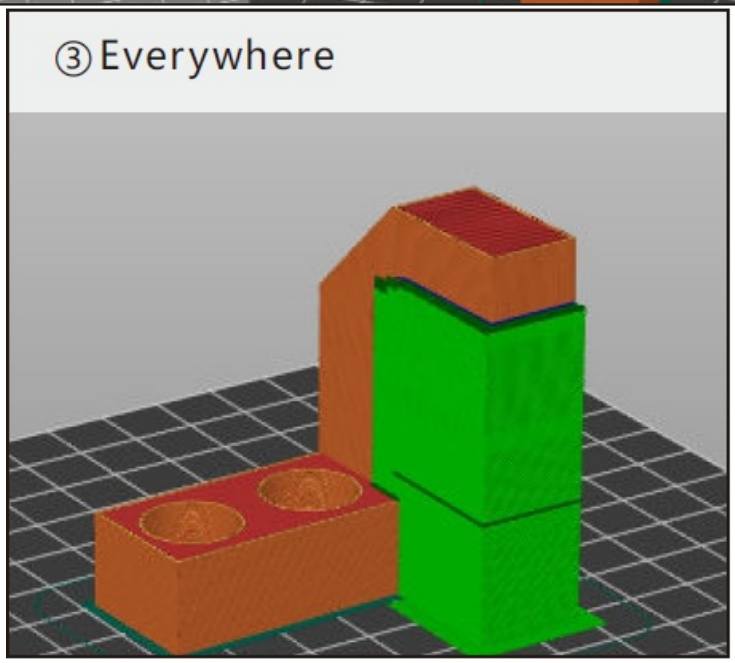


  - **Raft [10]:** Creates a raft below the part that can help with adhesion and provide a clean level surface to begin the print.
  - **Support type:** ① Support on the build plate only, ② For support enforcers only, ③ Everywhere

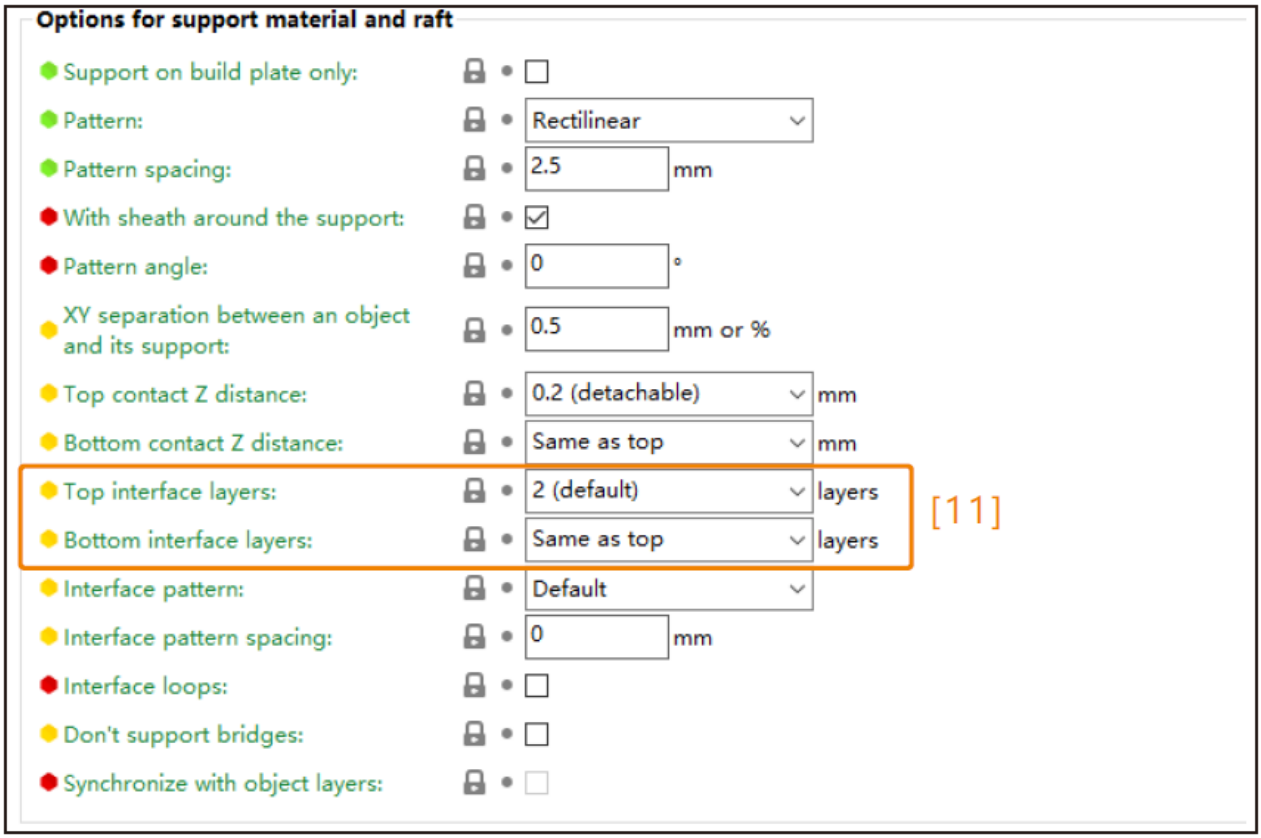


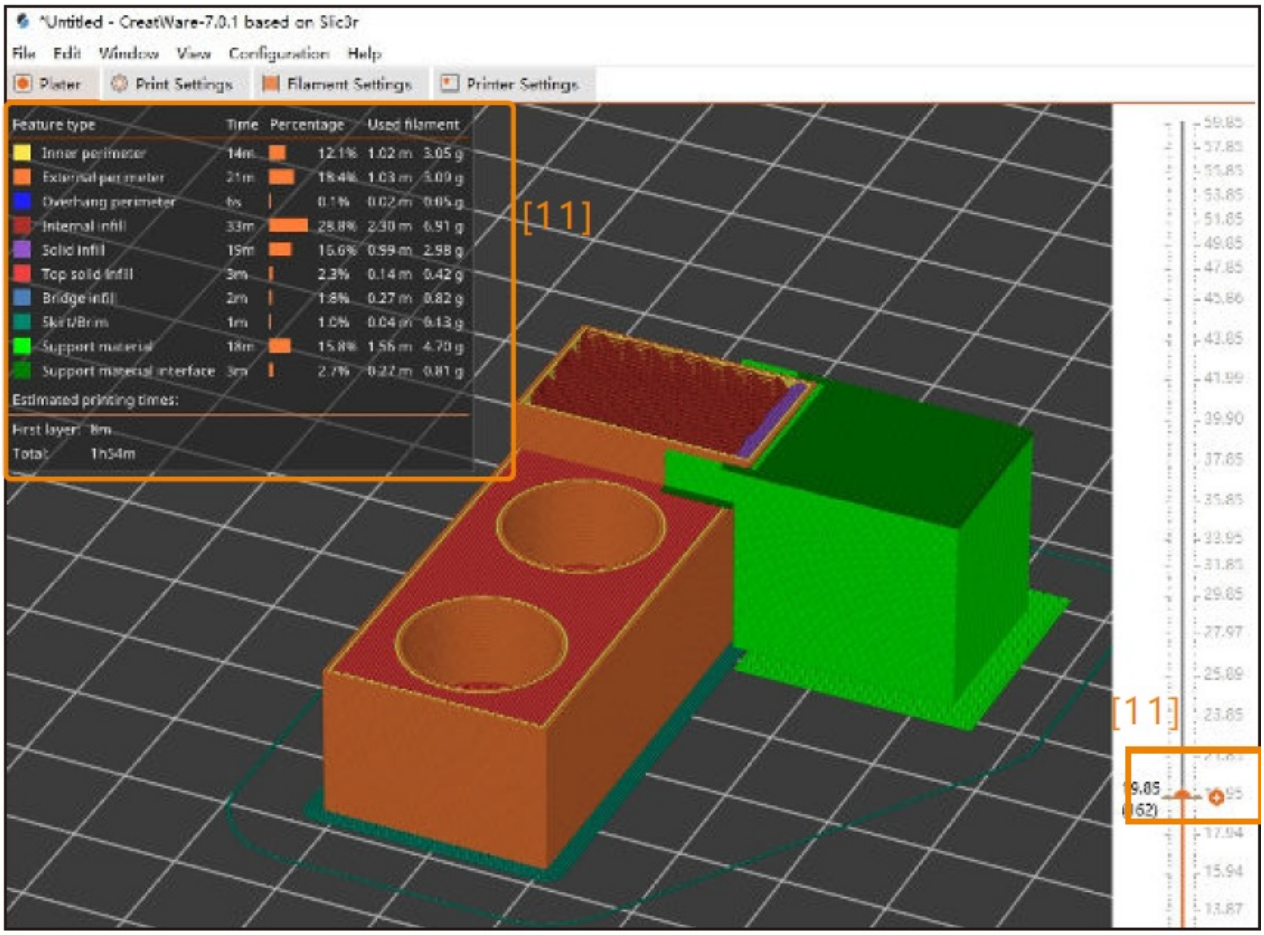






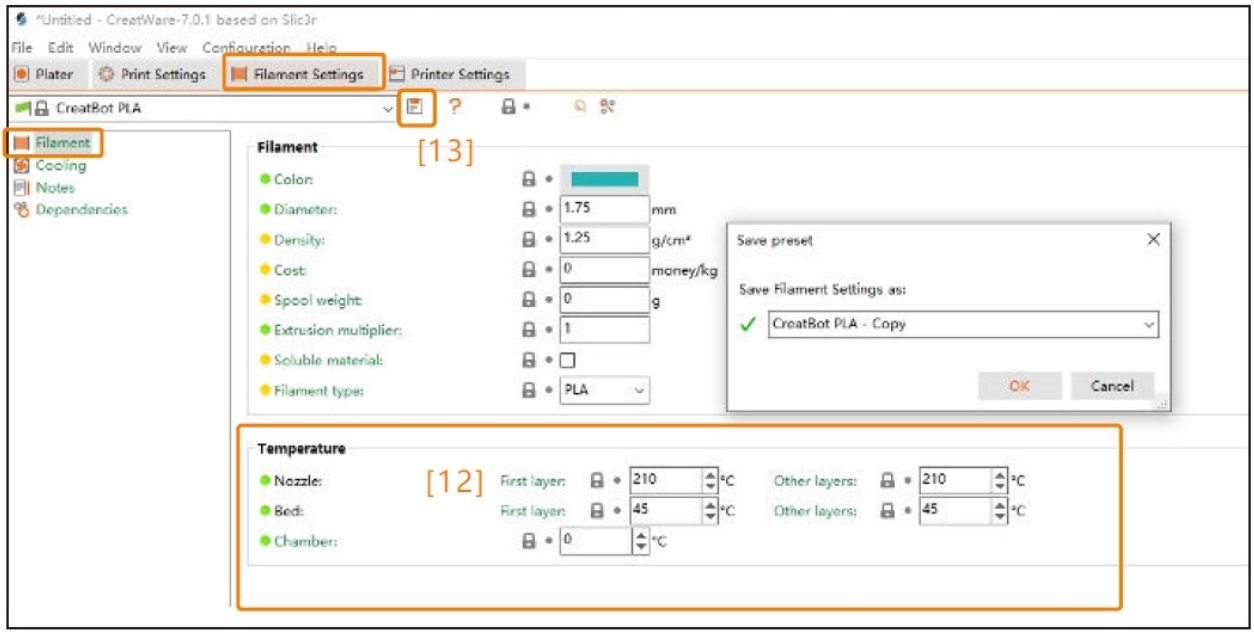
- **Top/Bottom interface layer:** The number [11] of layers of dense support. It will only exist in layers close to the surface of the model, printing a uniform surface between the object and the support material. This increases the contact area between the model and the support to prevent sagging and makes the support contact surfaces other after the support is removed. It is recommended to enable this feature.





- **Temperature**
  - **Temperature [12]:** Nozzle temperature refers to the melting [12] temperature of the filament, PLA is 210° C.
  - The hotbed temperature refers to the temperature of the heating plate during printing, and the PLA is 45°C.
  - Chamber temperature refers to the temperature of the printer chamber. PLA does not need this setting, it

only needs to be turned on when printing PC, PEEKand other materials. Refer to the filament temperature data sheet.



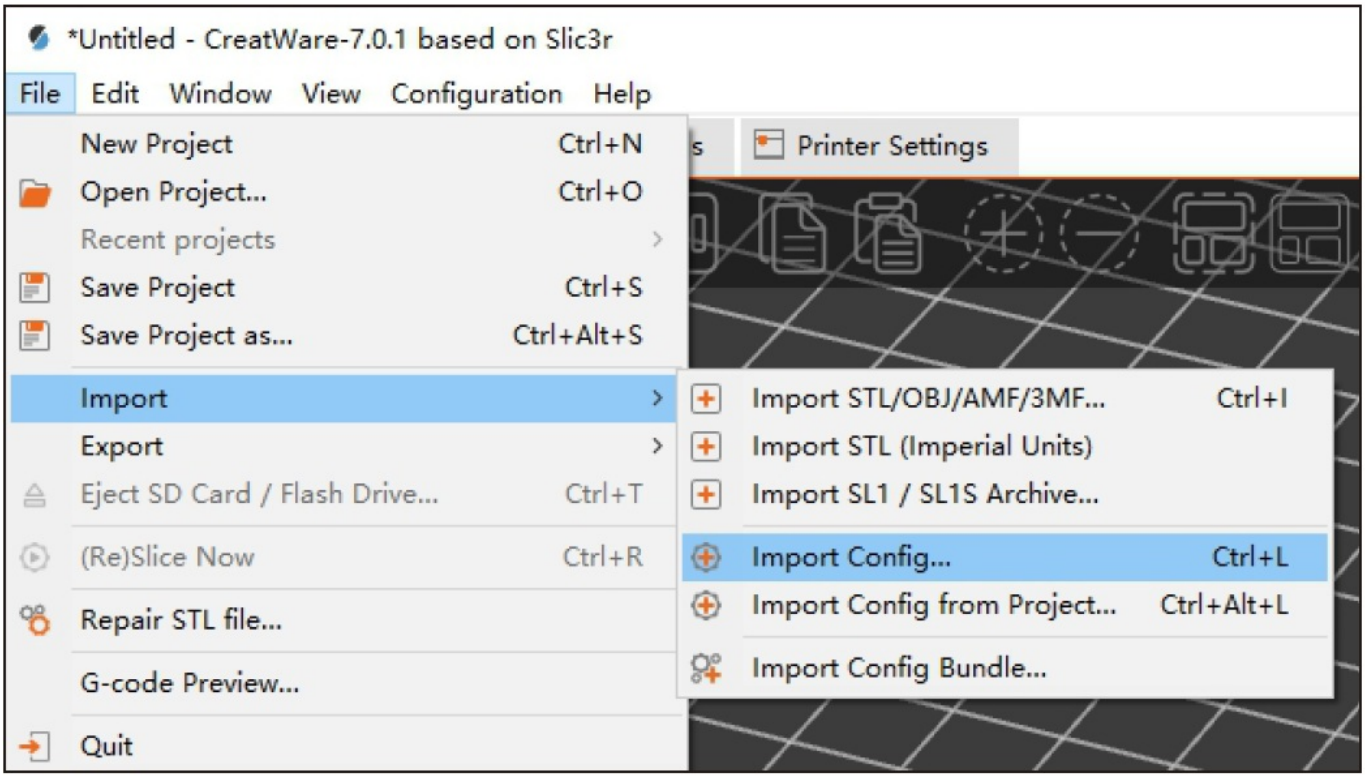
- **Save filament settings [13]:** If you want to use different filaments with the preset parameters, you can click this button to save the changed parameters and change the name.

Filament temperature datasheet

Parameter Filament	Print Temperature(℃)	Hot Bed ℃	Hot Chamber(℃)	Cooling Fan
PLAY	190-220	40-60	N/A	ON
ABS	240-260	80-100	N/A	LOW/OFF
TPU	210-230	40-50	N/A	ON
PETG	230-250	80	N/A	LOW/OFF
HIPS	230-250	110	N/A	OFF
ASA	260-270	90-110	50	LOW/OFF
PC	270-290	90-110	50	OFF
PAHT	280-320	70-80	N/A	LOW/OFF
PAHT-CF	300-320	70-80	N/A	OFF
PAHT-GF	280-320	70-80	N/A	OFF
S-Blue	280-300	60-120	N/A	OFF
S-PAHT	280-300	60-100	N/A	OFF
PEEK	400-450	120-150	70-120	OFF
PEEK-CF	400-450	120-150	70-120	OFF
PEEK-GF	400-450	120-150	70-120	OFF

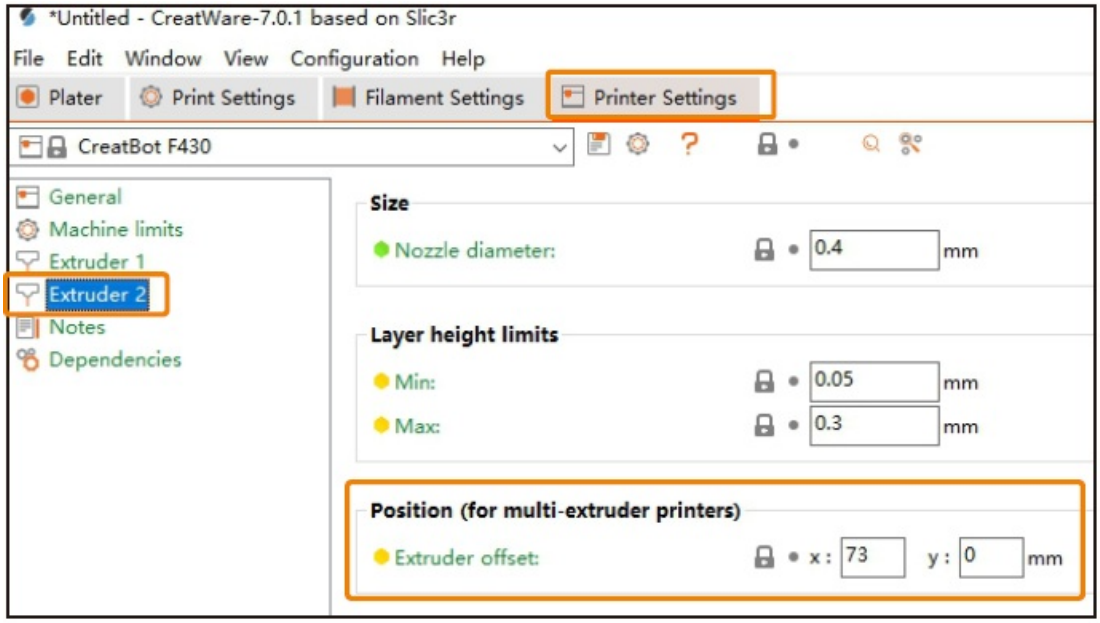
Import Config

Click File-Import-Import Config to quickly import the configured.

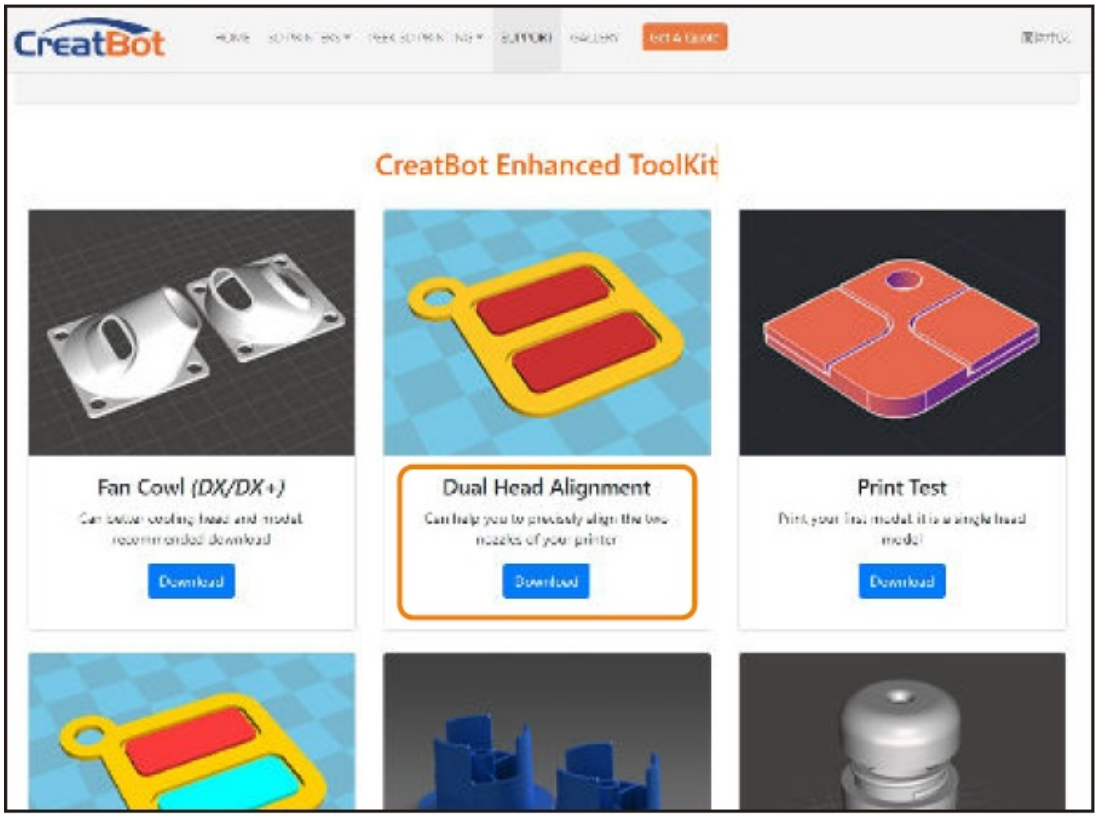


Dual-Extruder Print

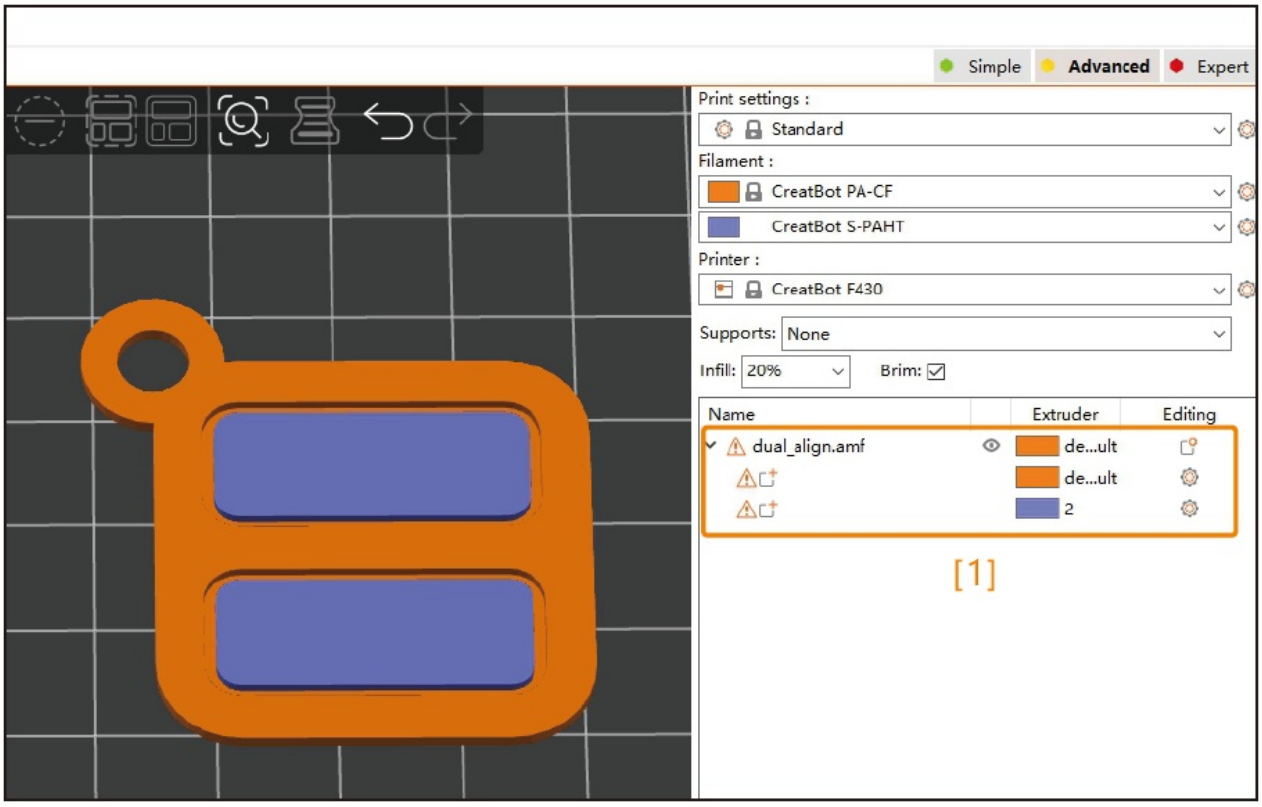
- **Calibrate X/Y Offset**
  - The printer's dual nozzles are offset, and the software has provided default values.
  - However, there is a slight error in the assembly process of the nozzle, so the color of the model may not be aligned when printing. At this time, we need to recalibrate the offset value of the nozzle.
  - For example, the default nozzle offset value of F430 is X73, Y0.



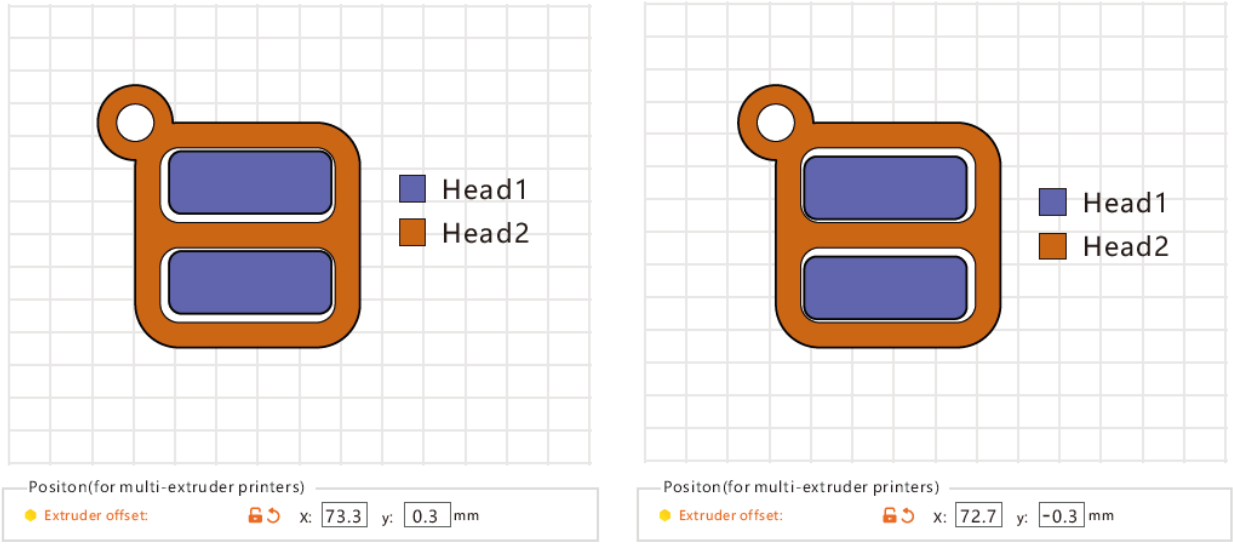
  - First, download the dual head alignment model [Download-CreatBot](#).



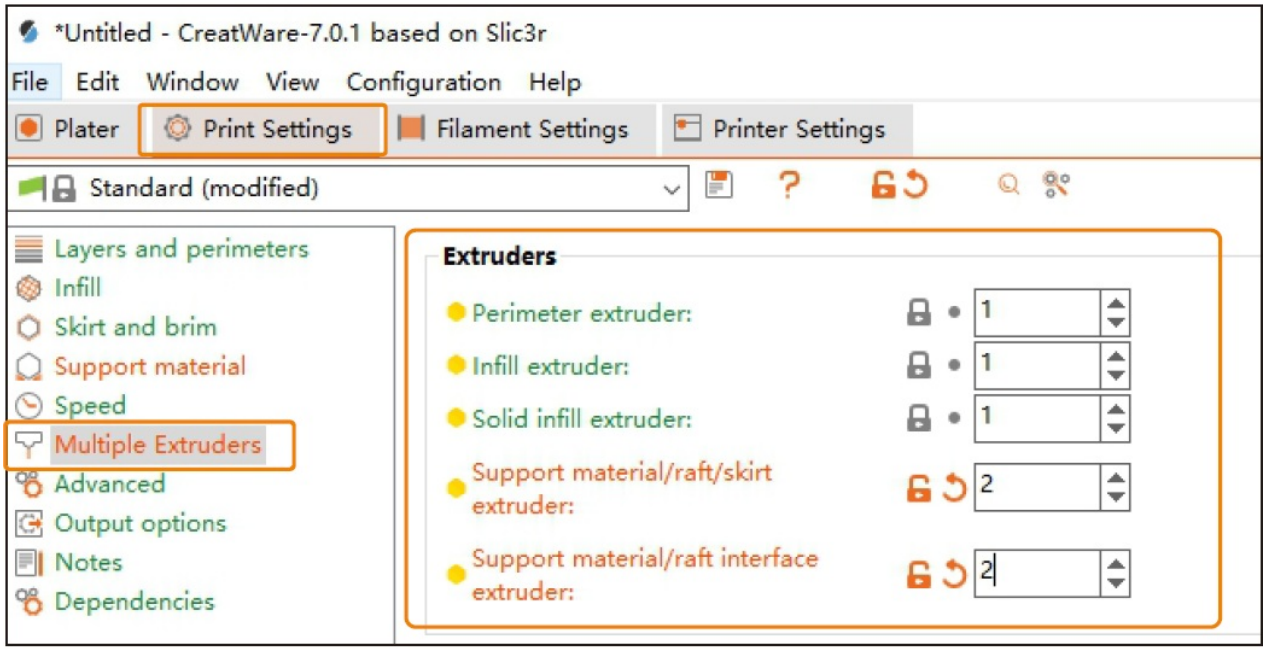
- Click the extruder icon [1] to define the number of [1] extruders, then slice and print.



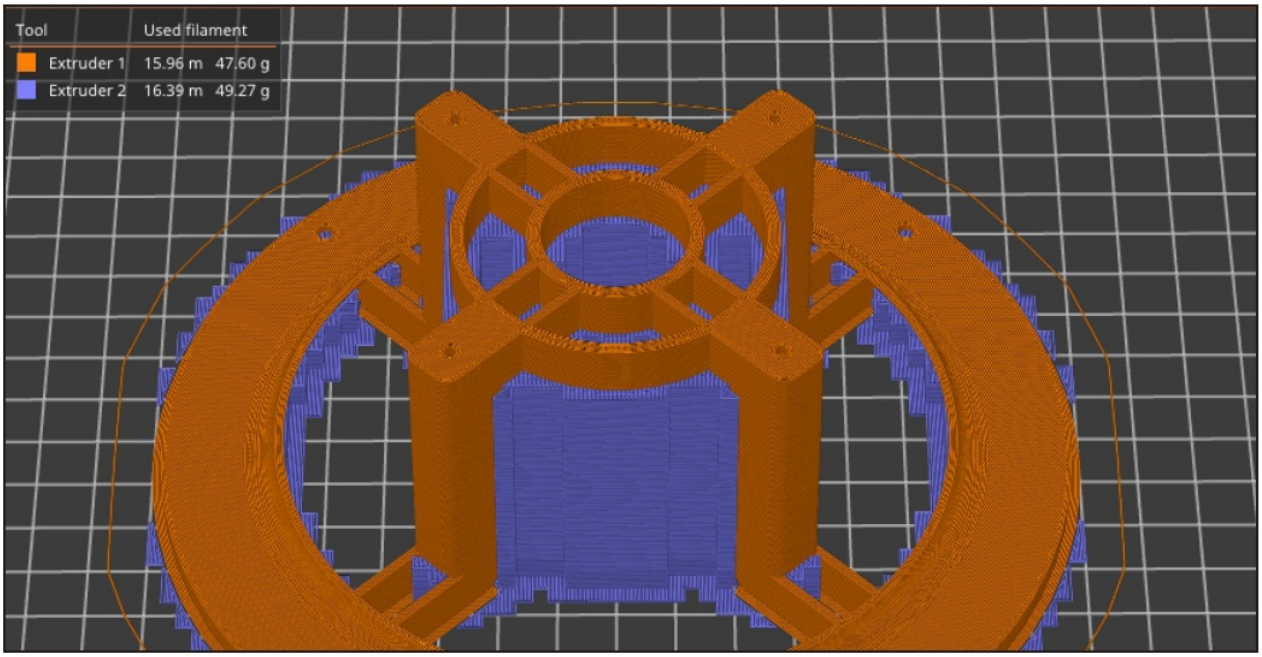
- When printing the result as shown below, the offset value should be adjusted to X73.3, Y0.3
- When printing the result as shown below, the offset value should be adjusted to X 72.7,Y-0.3



- **Main object+support**
  - Select perimeter/infill/solid infill to use extruder 1, and support/raft/brim to use extruder 2.

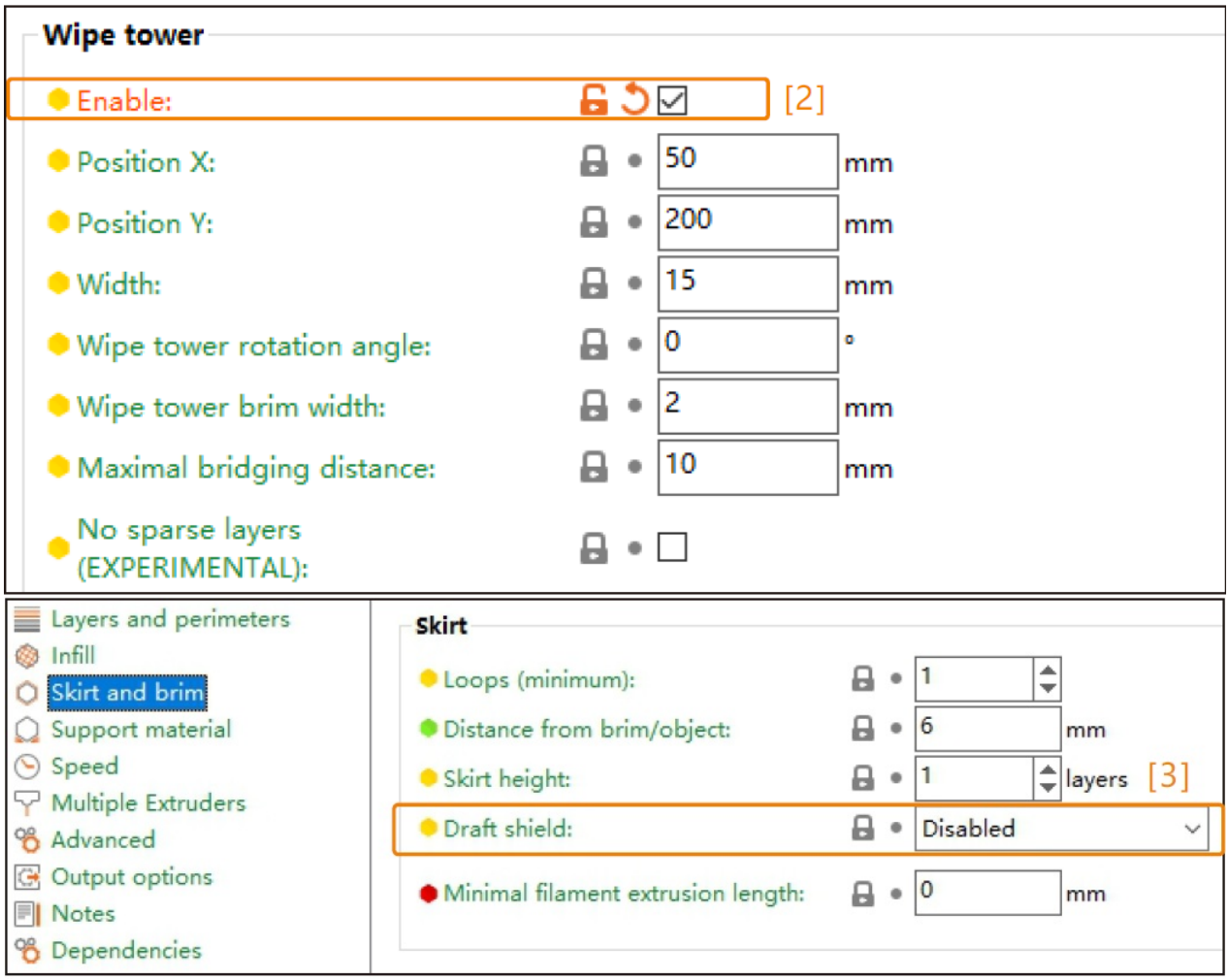


  - In the preview interface, you can view the weight of the main object and support filaments.

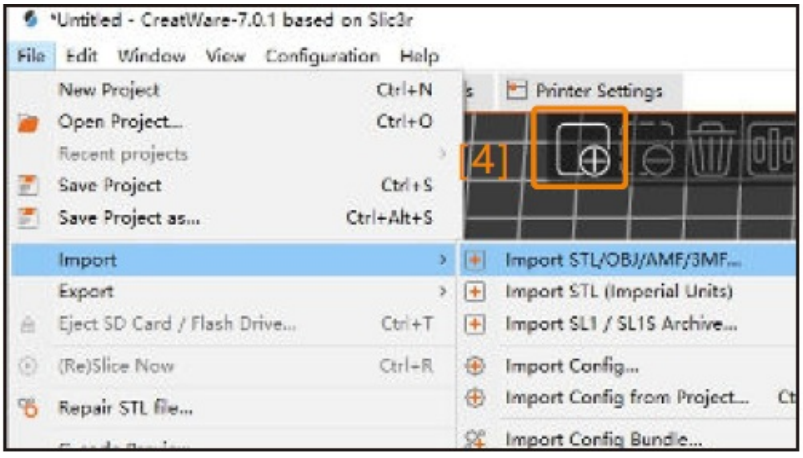


  - When printing with dual extruders, to catch the extra oozing from the idle extruder, it is recommended to use the wiper tower [2] (Print Settings – Multiple Extruders – Wipe Tower) or use the draft shield [3] (Print

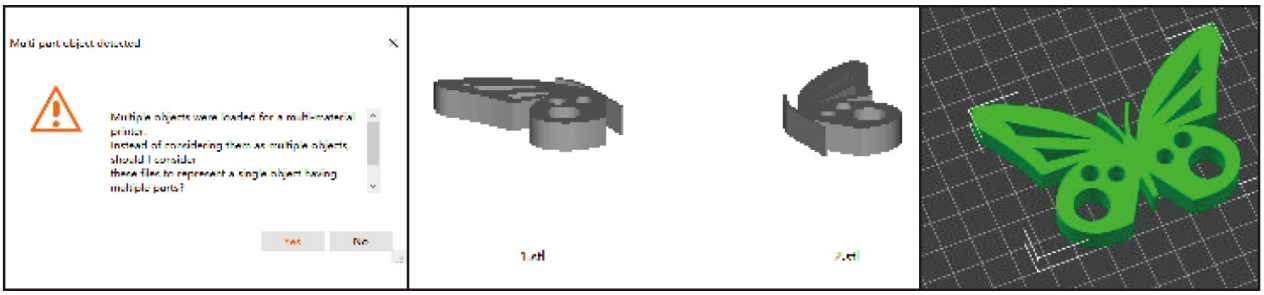
Settings – Skirt and brim – Skirt – Draft shield).



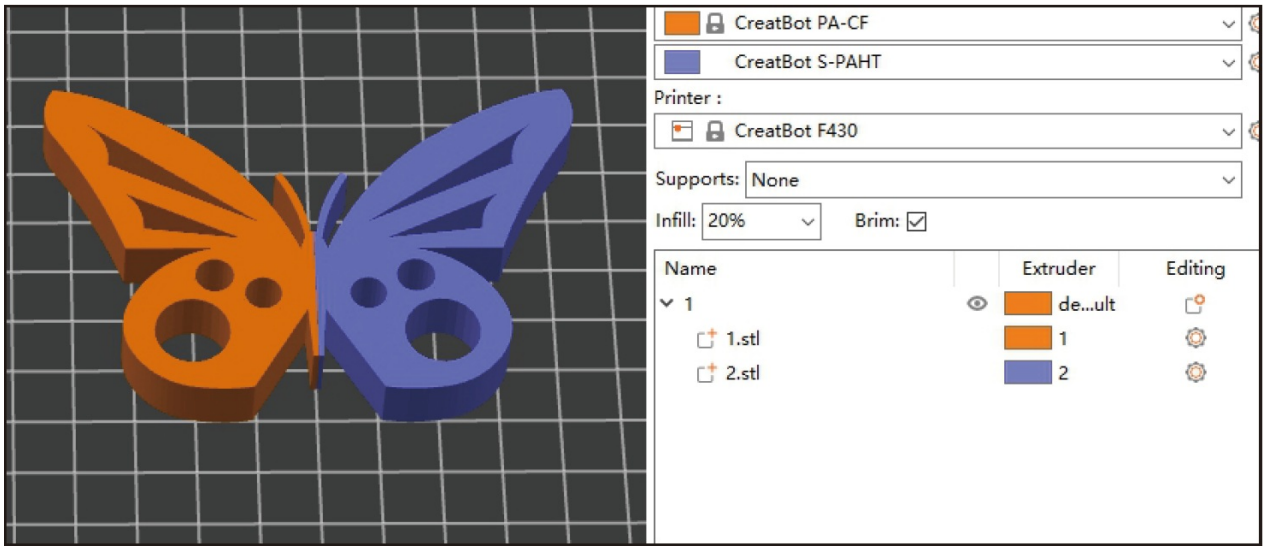
- **Dual-Color Printing**
  - The files must be two models in the same coordinate system.
  - **There are three ways to import files:**
    1. Drag all STL files into CreatWare at the same time
    2. Select File – Import and select all STL files at once
    3. Click the “Add” [4] button to select all STL files at once.



  - After importing the model, click Yes, and the two models will be automatically merged.



- Set the number of nozzles to be used for each part separately, and install different color filaments on the printer to print.

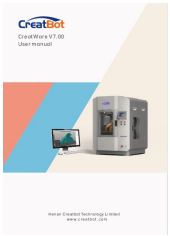


CONTACT INFORMATION

- Henan Creatbot Technology Limited
- **Tel** 0371-55691230
- **Skype ID:** Creation
- **Website** www.creatbot.com
- **Email** sales@creatbot.com support@creatbot.com
- **Address** 6#, Chaoya Industry Park,
- Hanghai Road, No.13 Ave.,
- Eco-Tech Development Zone, Zhengzhou City,
- Henan Province, China
- **We Chat scan**

- Henan Creatbot Technology Limited www.creatbot.com.

Documents / Resources

	**CrEatBot D600 Professional Large Volume 3D Printer** [pdf] User Manual D600 Professional Large Volume 3D Printer, D600, Professional Large Volume 3D Printer, Large Volume 3D Printer, Volume 3D Printer, 3D Printer, Printer

References

- large-scale & High performance 3D printers | CreatBot
- User Manual