CreatBot

CreatWare V7.00
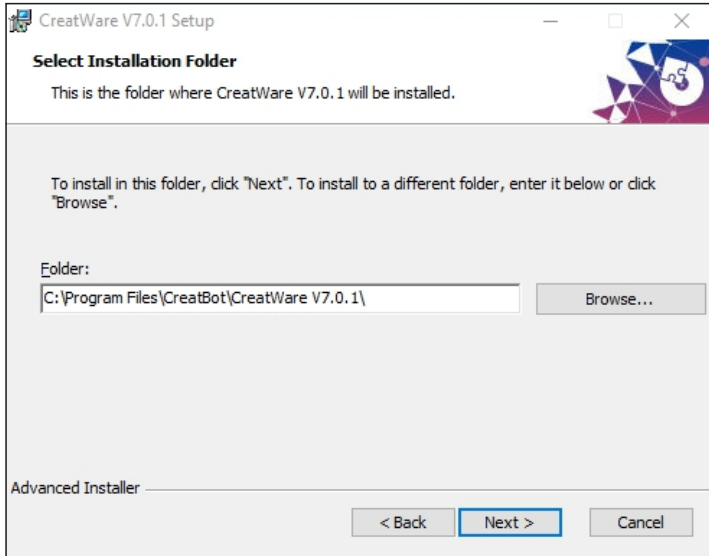## User manual

CONTENTS
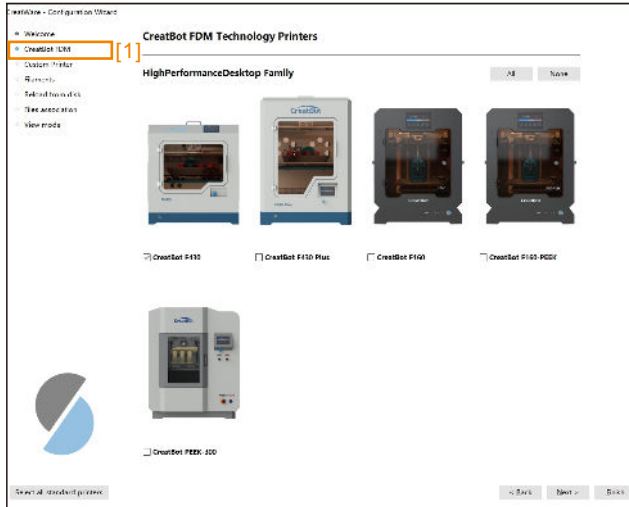
1. CreatWare Installation

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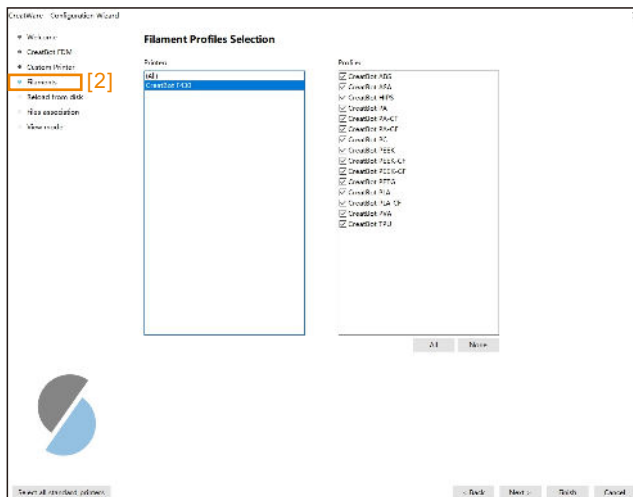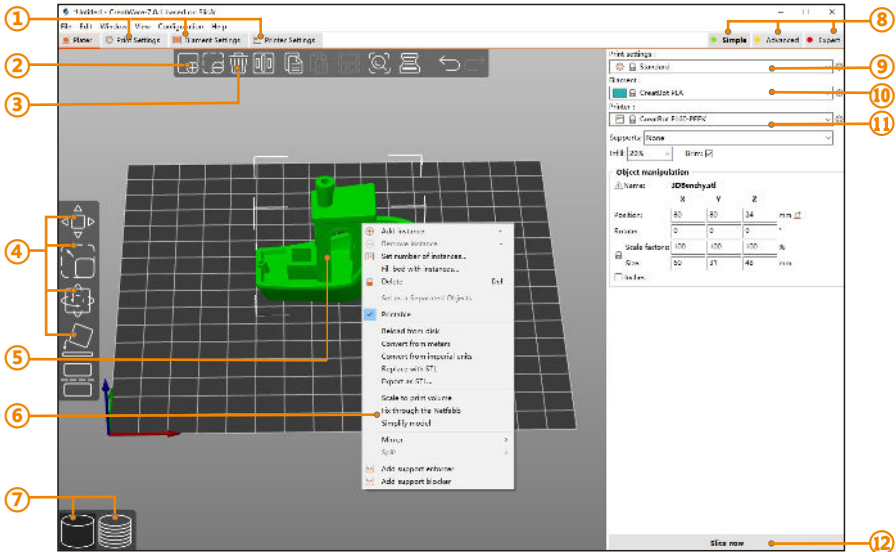
2. Configuration Wizard

CreatBot FDM [1] : Select your 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 similar filament first, then change settings in the software, save.

3. 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 model opens a context menu

⑦   Switch between 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
B	Zoom on the entire print bed	Z	Zoom on selected objects or on all objects if none selected
🖱️ / O	Zoom out	🖱️ / i	Zoom in
0	Isometric view	1	Top-down view
2	Bottom-up view	3	Front view
4	Back view	5	Left view
6	Right view		

CreatBot 3D Printer

4. Import Model

Import model

Click the "Add" (Ctrl+I) button to import the models. (stl,obj,amf files)
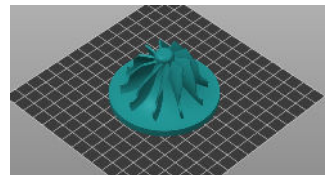
Arrange

Click the "Arrange" (Shift+A) button to auto-arrange the models.
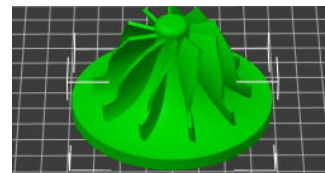
Scale

Click "Zoom" (S), you can see a contour line around the model. Drag the outline to adjust 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.
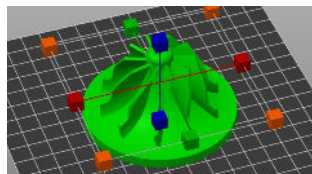
Rotate

Click"Rotate"(R), you can see three ting line around the model, respectively along the X,Y,Z direction. Drag the pointer to rotate the desired angle. You can also enter a value in the lower right corner.
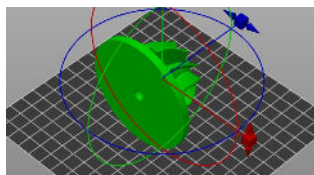
Place on face

Click"Place on face(F)", multiple white planes will appear on the model. Clicking on any of the white planes will align that plane with the print bed.
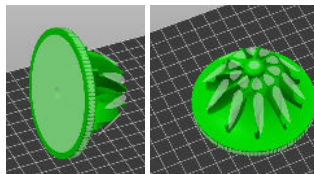
5. Slicing Operation

5.1 Parameter Modes

🟢 Simple mode	🟡 Advanced mode	🔴 Expert Mode
Simple mode: only display the basic parameters, suitable for beginners to print common material.	Advanced mode : display more parameters, suitable for advanced users.	Expert Mode: display all settings parameters, you can customize the printer parameters number, G-code, etc.

5.2 Slicing Step
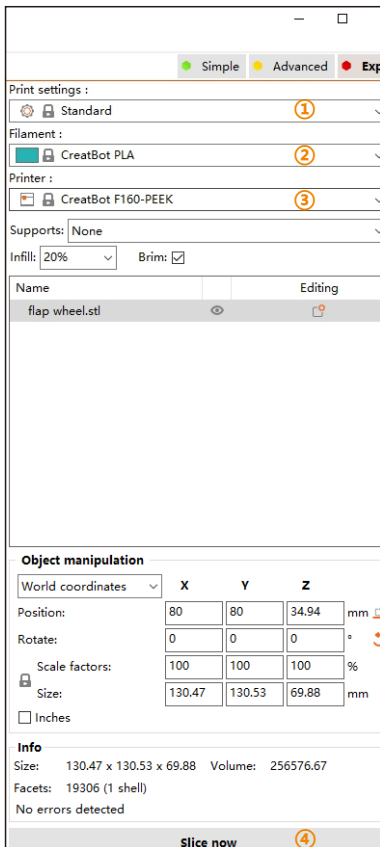
Step 1: Select print settings. Three print settings are provided by default, Standard, Quality, Speed. You can also change each setting according to your needs.
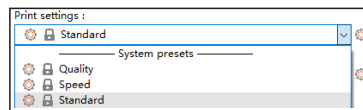
Step 2: Select filament.
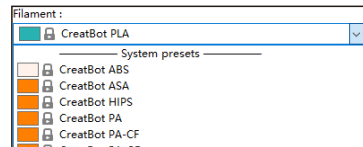
Step3: Select printer.
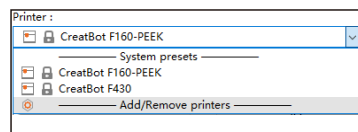
Step4: Click "slice now"

6. 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.
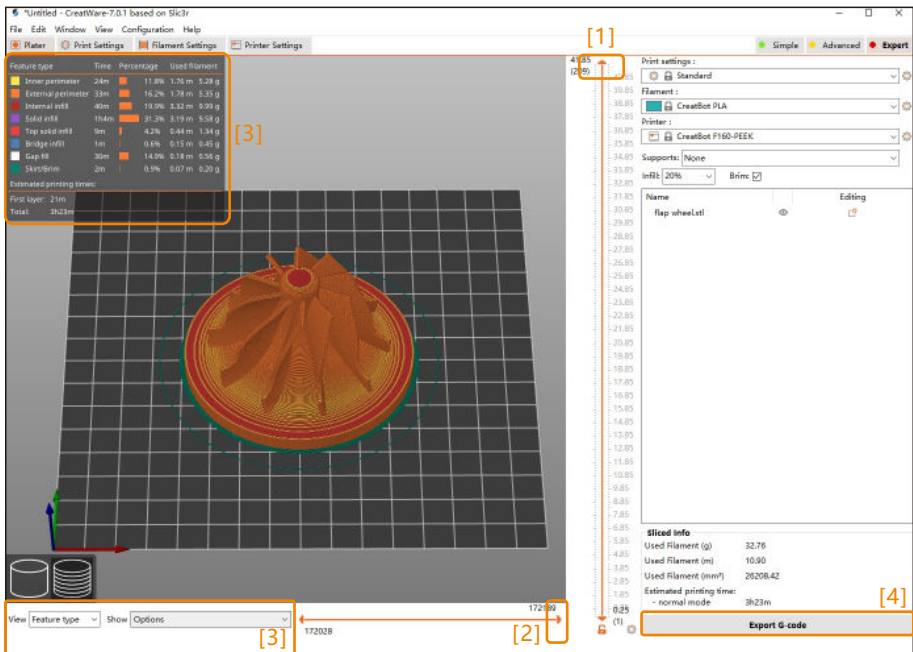
Export G-code: Click Export G-code[4], save the G-code to the U disk, insert the U disk into the printer port, select the file and start printing.

7. Common Settings

7.1 Extruder

7.1.1 Nozzle diameter[1]: The default size is 0.4mm. If you use a large size nozzle, the extrusion of the filament will multiply and need to increase the printing temperature.
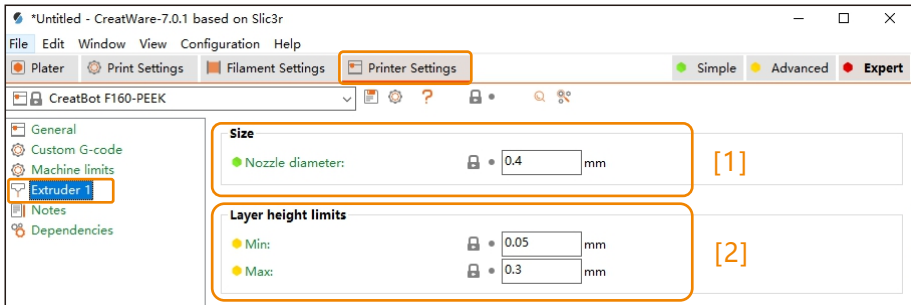
7.1.2 Retraction [2]: Direct drive extruder (such as F430, PEEK-300, D600Pro), the default retraction distance is 2mm (within 3mm); Bowden extruder (such as DX, DE), the default retraction distance is 4 -5mm. If the retraction distance is too small, will leave some strings, you can increase the retraction distance. If the retraction distance is too large, the printing time will be affected.

Strings picture:
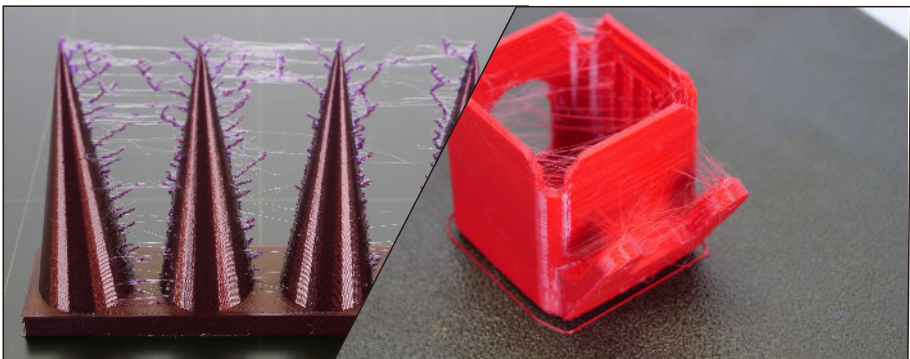
7.2 Layer

7.2.1 Layer height[3]: The most important parameter 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.
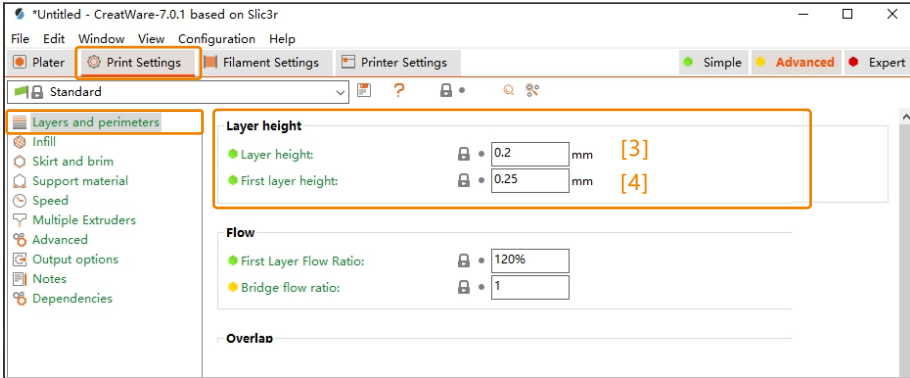
7.2.2 First layer height[4]: It refers to the height of the first layer of printing, generally greater than 0.2mm. This parameter can compensate the errors of the platform and make the model stick to the platform better.

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

7.3 Speed

Speed[5]: Speed affects the surface quality of the 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.
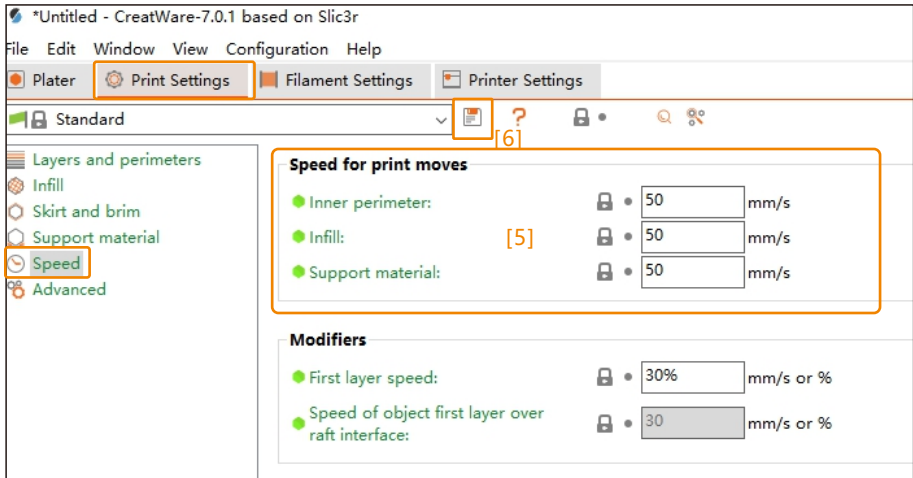
Change printing settings[6]: After changing the printing parameters, click save and customize the name for future use.
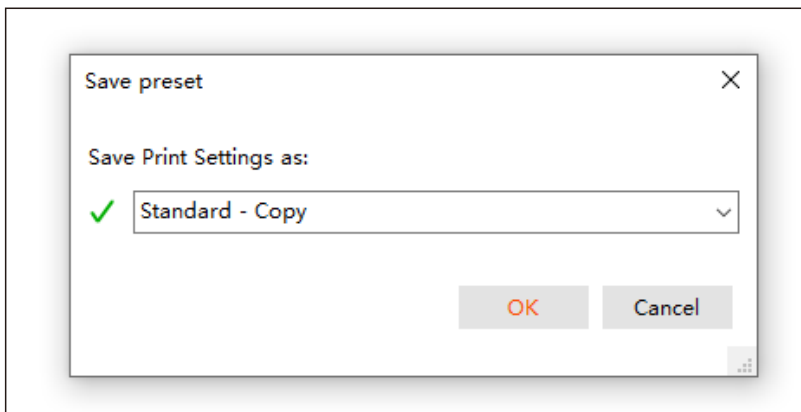
7.4 Skirt and Brim

7.4.1 Skirt[7]: It is mainly used to confirm the printing area and check whether the nozzle is extruded normally.

7.4.2 Brim[8]: Print a series of concentric lines around the model to increase the adhesion between the model and the platform.
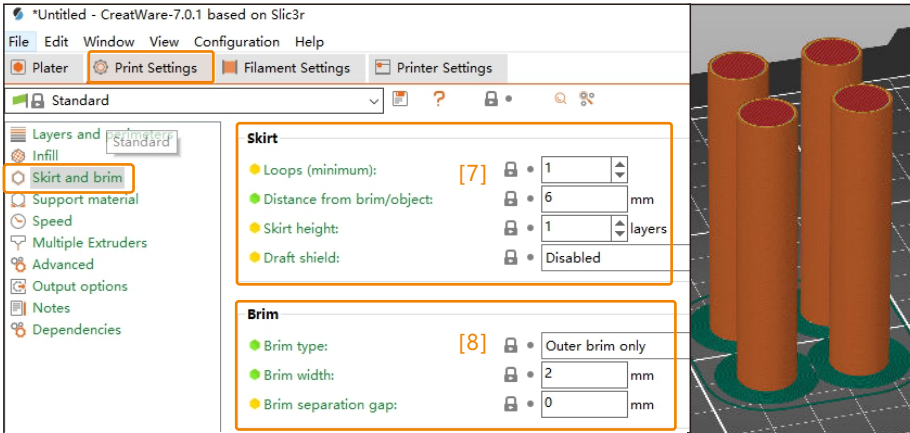
7.5 Support

7.5.1 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.
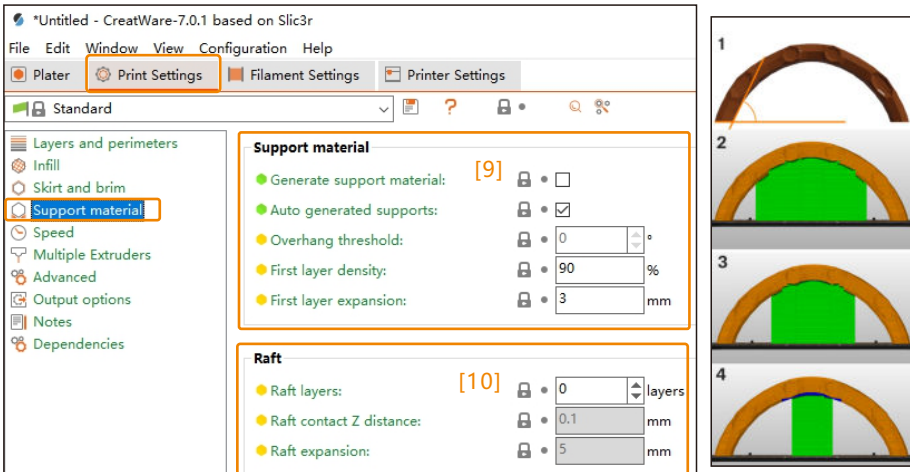
7.5.2 Raft[10]:Creates a raft below the part that can help with adhesion and provide a clean level surface to begin the print.

7.5.1 Support type: ① Support on the build plate only, ② For support enforcers only, ③ Everywhere

① Support on the build plate only
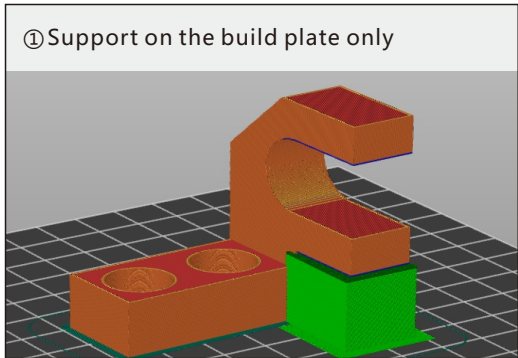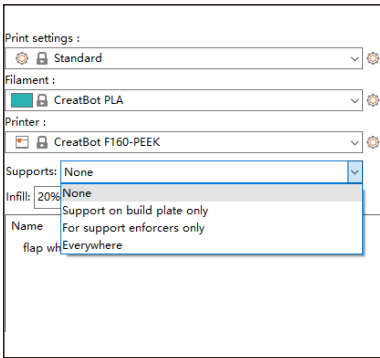
② For support enforcers only : Left-click to paint the position where support needs to be added.
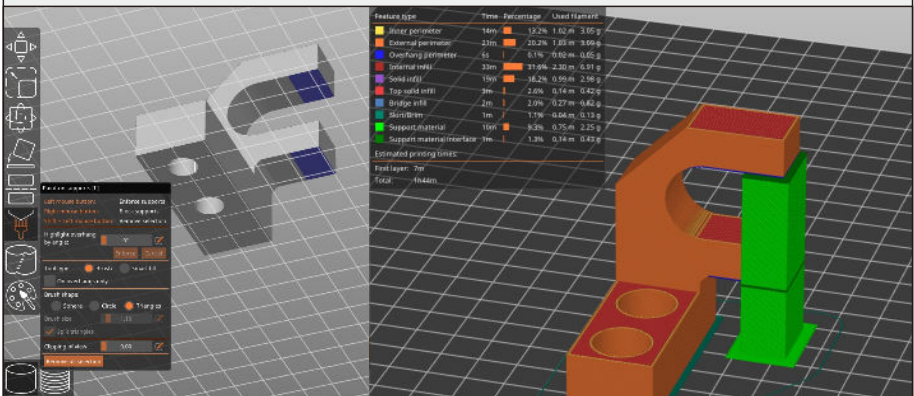
③ Everywhere
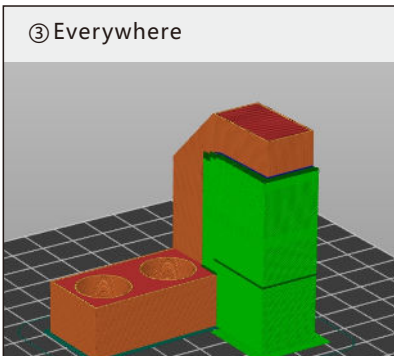
7.5.4 Top/Bottom interface layer[11]: The number of layers of dense support. It will only exist in layers close to the surface of the model, printing a uniform surface between the object and the support material. This increases the contact area between the model and the support to prevent sagging, and makes the support contact surface smoother after the support is removed. It is recommended to enable this feature.

7.6 Temperature

Temperature[12]: Nozzle temperature refers to the melting 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, PEEK and other materials. Refer to the filament temperature data sheet.
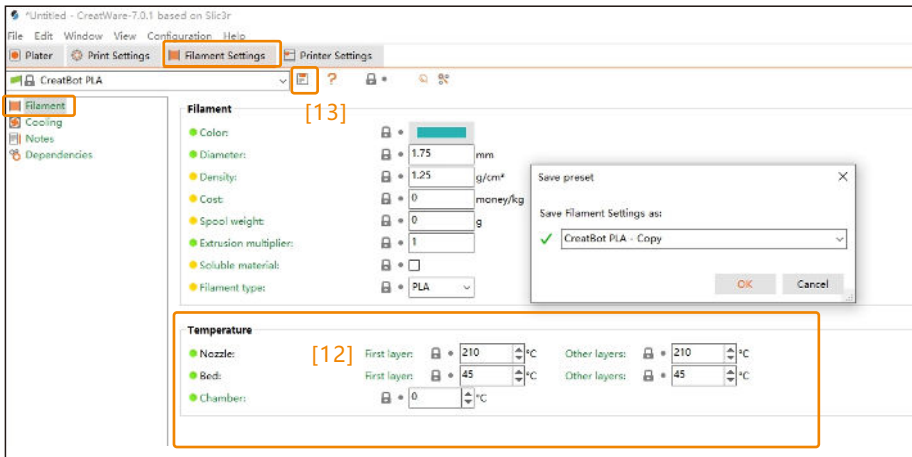
Save filament settings[13]: If you want to use different filaments with the preset parameters, you can click this button to save the changed parameters and change the name.

Filament temperature data sheet

Parameter / Filament	Print Temperature(℃)	Hot Bed（℃）	Hot Chamber(℃)	Cooling Fan
PLA	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

8. Import Config

Click File-Import-Import Config to quickly import the configured.
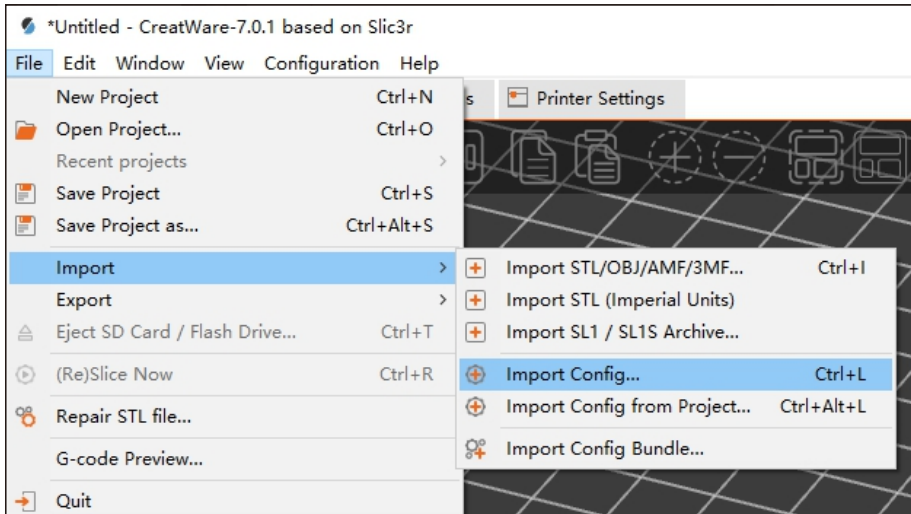
9. Dual-Extruder Print

9.1 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.
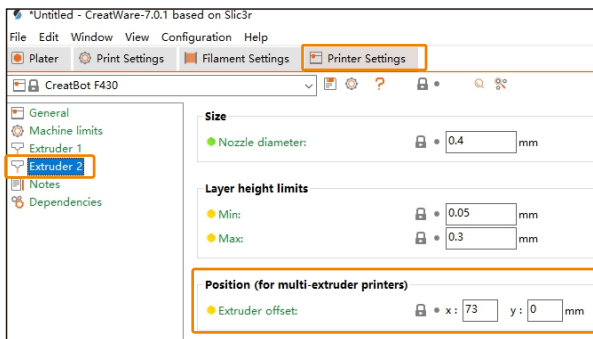
First, download the dual head alignment model   Download-CreatBot
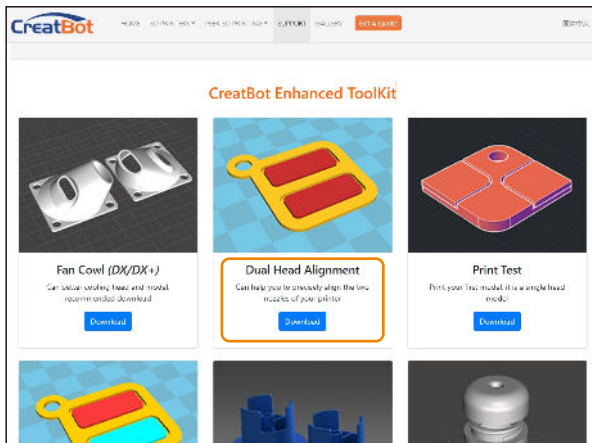
Click the extruder icon [1] to define the number of extruder, then slice and print.
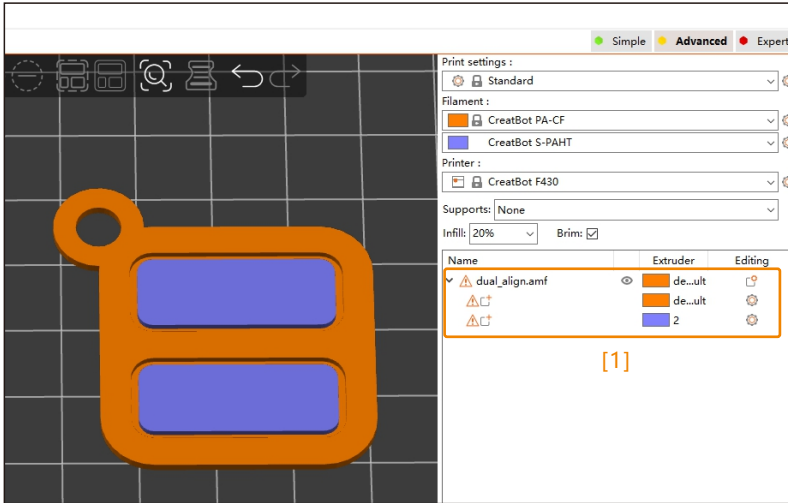
When printing the result as shown below, the offset value should be adjusted to X73.3 , Y0.3
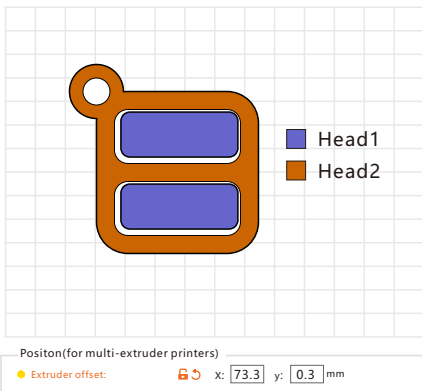
When printing the result as shown below, the offset value should be adjusted to X 72.7 ,Y-0.3
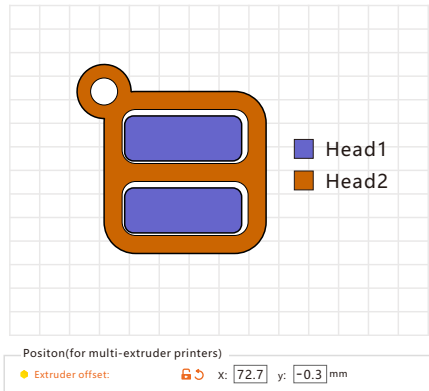
9.2 Main object+support

Select perimeter/infill/soild infill to use extruder 1, 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, in order to catch the extra oozing from 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).
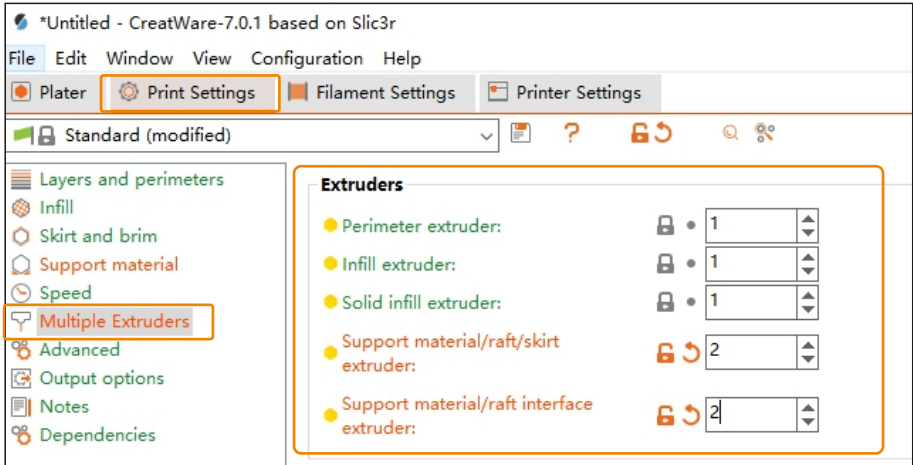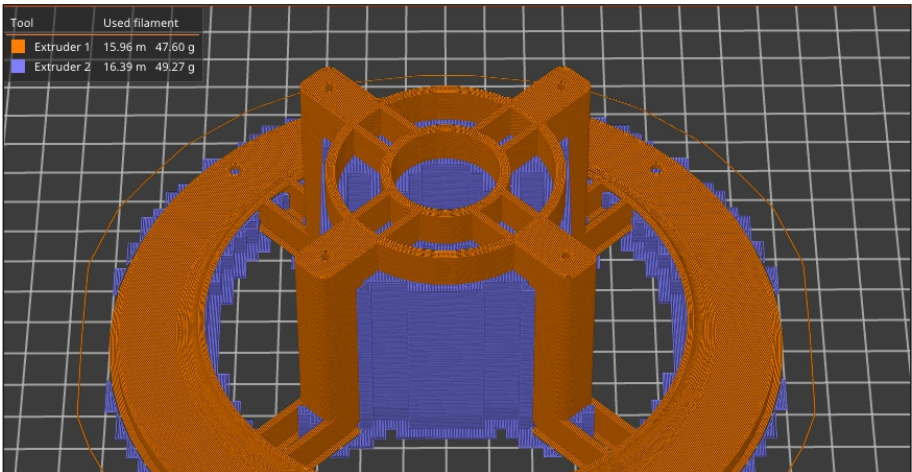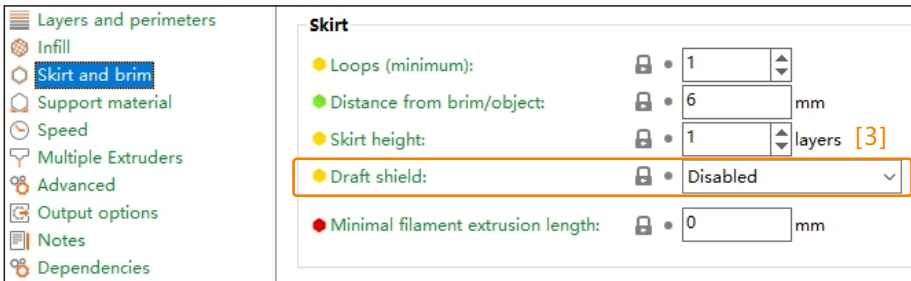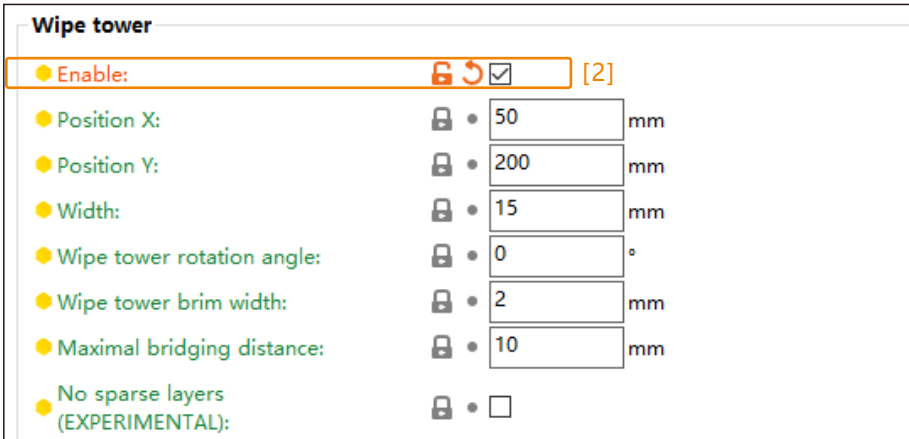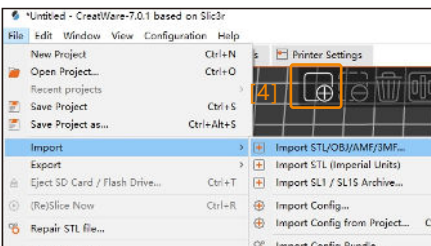
Wipe tower

● Enable:	🔓 ↻ ☑	[2]
● Position X:	🔒 • 50	mm
● Position Y:	🔒 • 200	mm
● Width:	🔒 • 15	mm
● Wipe tower rotation angle:	🔒 • 0	°
● Wipe tower brim width:	🔒 • 2	mm
● Maximal bridging distance:	🔒 • 10	mm
● No sparse layers (EXPERIMENTAL):	🔒 • ☐	

Layers and perimeters
Infill
Skirt and brim
Support material
Speed
Multiple Extruders
Advanced
Output options
Notes
Dependencies

Skirt

|		
● Loops (minimum):	🔒 • 1	
● Distance from brim/object:	🔒 • 6	mm
● Skirt height:	🔒 • 1	layers [3]
● Draft shield:	🔒 • Disabled	
● Minimal filament extrusion length:	🔒 • 0	mm

9.3 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 together.
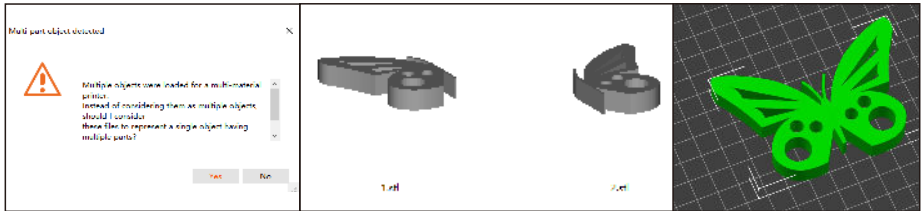
Set the number of nozzle to be used for each part separately, and install different colors filaments on the printer to print.
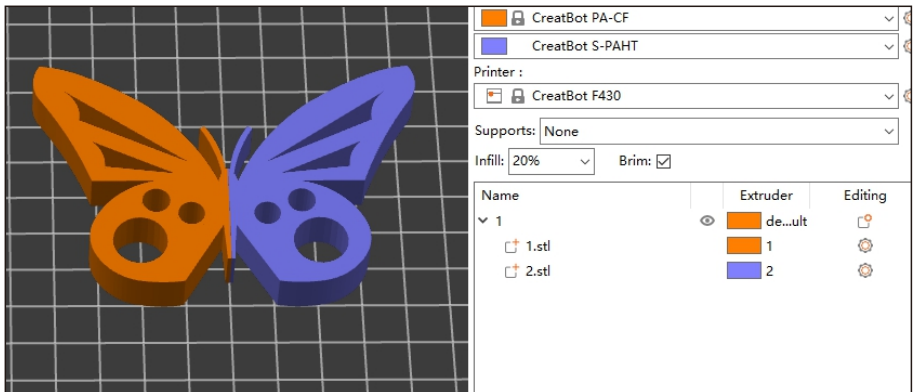
Quality · Experience · Service

Henan Creatbot Technology Limited

Tel：0371-55691230

Skype ID：creatbot

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

WeChat scan