How to Print by Couple

Belindon Technology co., Ltd

Note: The firmwares of printing dual color and mixed color are different.
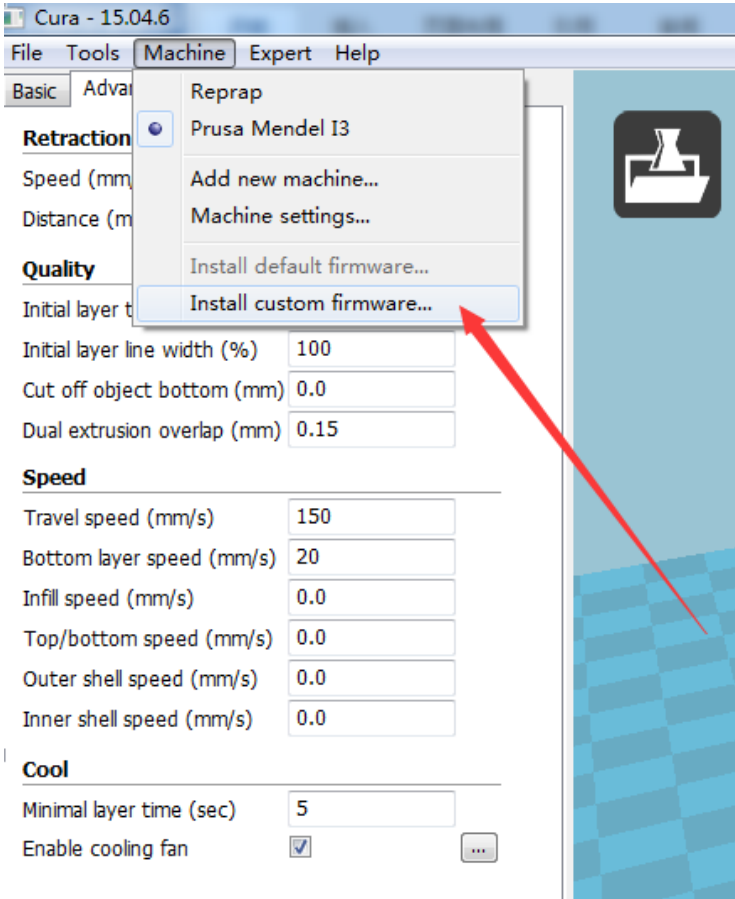
5.1 Printing Dual Color

5.1.1 Machine set-up

Belt is tight enough---screw is tight enough---the wire connection is correct---x/y/z axis movement is correct--- auto home works fine--- temperature works fine---preheat works fine.

5.1.2 Upload Firmwares

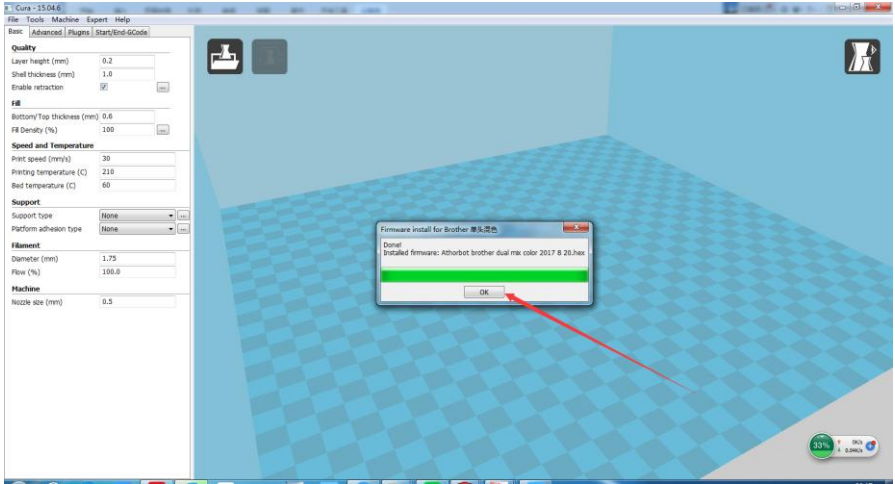Connect your computer and the 3D printer control box with the USB cable.

Open Cura

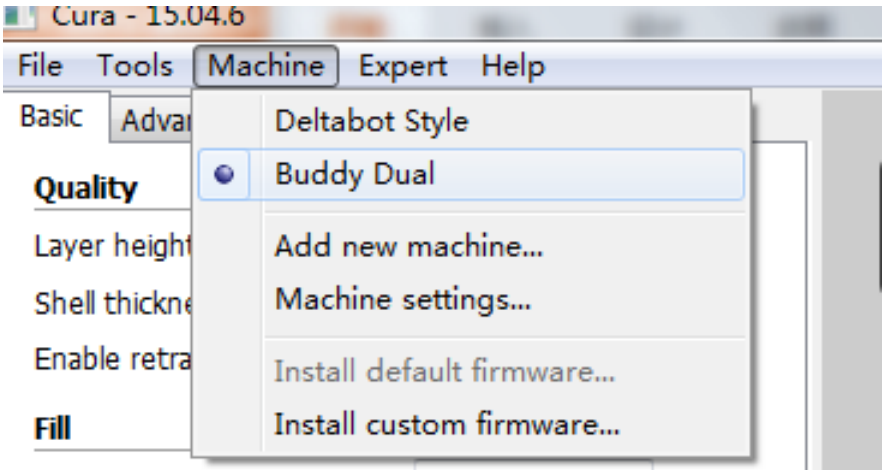Load the firmware which in .hex format. (VIP member can download the latest firmware
http://members.athorbot.com/VIP/

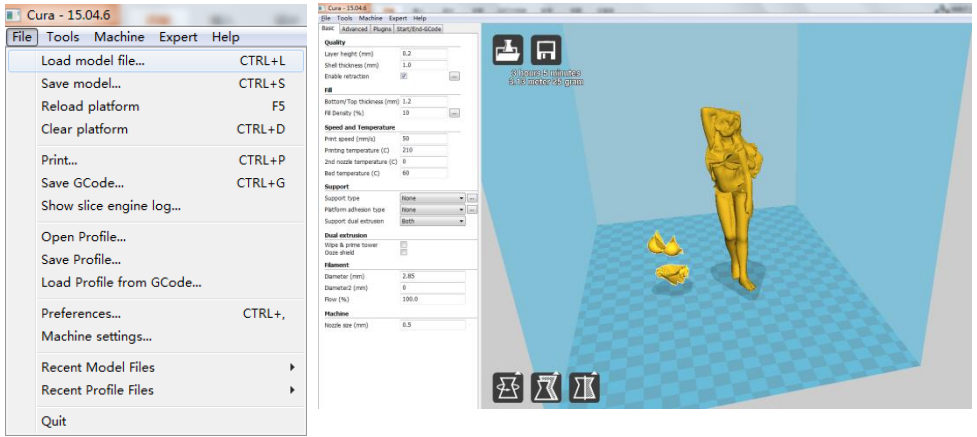eg.:   3,1,2 Couple(Buddy)Dual Color v1.0.hex

Well Done! Enjoy your printing journey!

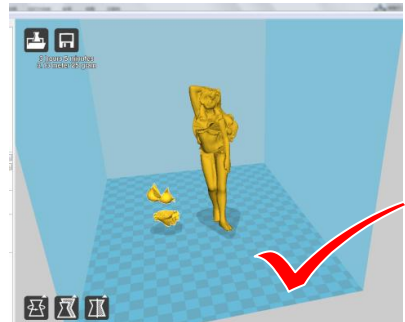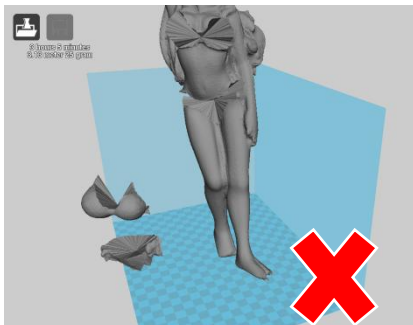## 5.1.3 Slicing

Select the machine to print dual color print

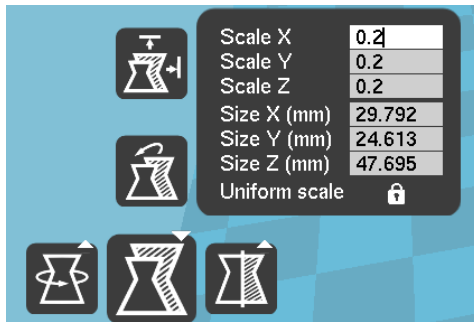Click "File" and select "Load model file " to upload models (*STL formats) as pictures below:
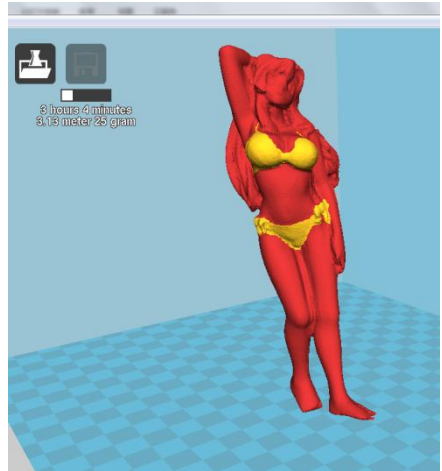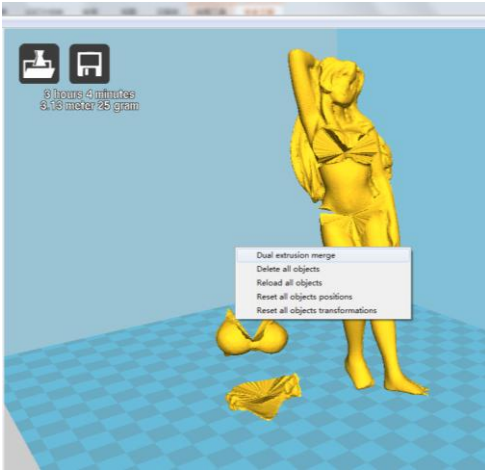

Please make sure that the model is inside the "blue cuboid". If the size of model is too large, please click" ⧗ " at the left bottom to scale the model.
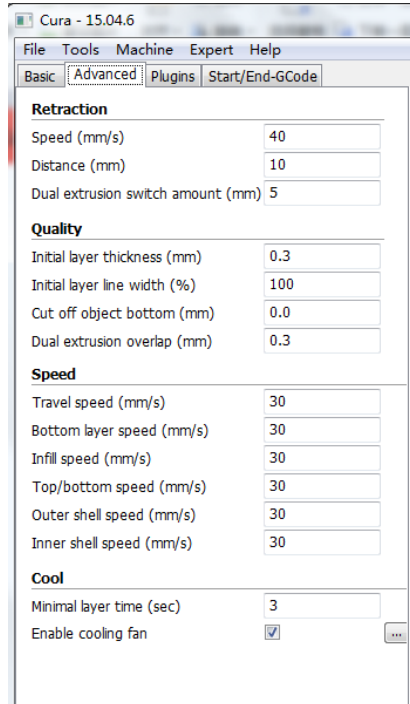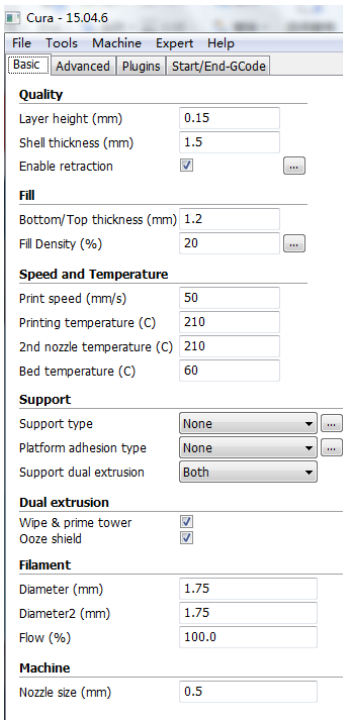
Right click and choose "Dual extrusion merge" to synthesize two models as the following picture
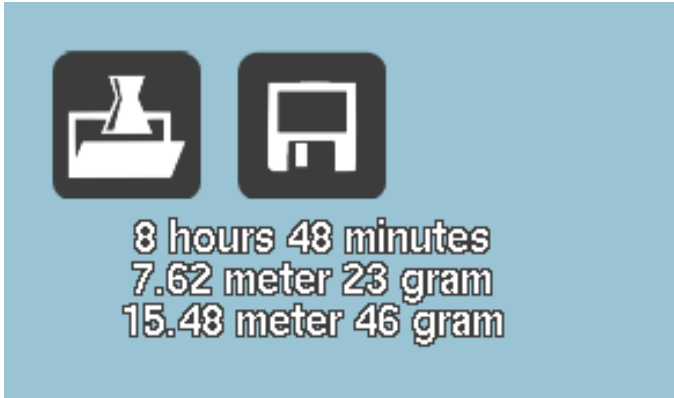

Slicing Setting (for example: Print PLA filament)

When finish slicing setting, the printing time will appear in the top left of the window. (PS: The length of printing time depends on the size of model as well as parameter setting.

8 hours 48 minutes
7.62 meter 23 gram
15.48 meter 46 gram

Saving files (Format: *Gcode)

Right click " 💾 " and choose "Save Gcode", or click "File" → "Save Gcode ".

Save toolpath

8 hours 48
7.62 meter 23 gram
15.48 meter 46 gram

Save GCode...
Slice engine log...

Cura - 15.04.6

| File | Tools | Machine | Expert | Help |

Load model file...   CTRL+L
Save model...   CTRL+S
Reload platform   F5
Clear platform   CTRL+D

Print...   CTRL+P
Save GCode...   CTRL+G
Show slice engine log...

Open Profile...
Save Profile...
Load Profile from GCode...
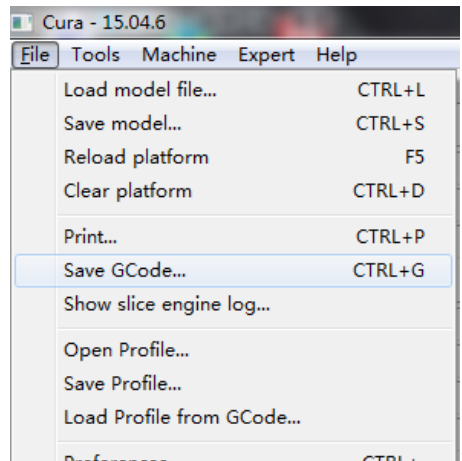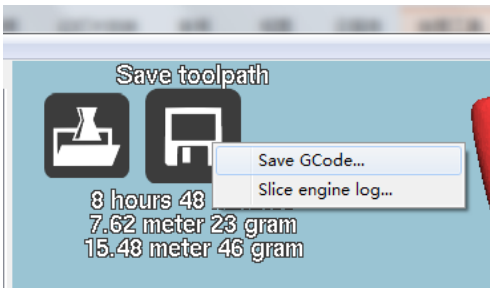
5.2 Print from SD card

Insert SD Card with .gcode file in the card slot provided in Athorbot control box.

Follow these settings on your 3D Printer LCD control system

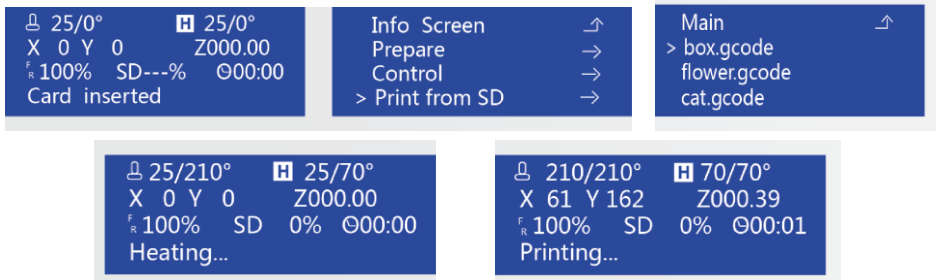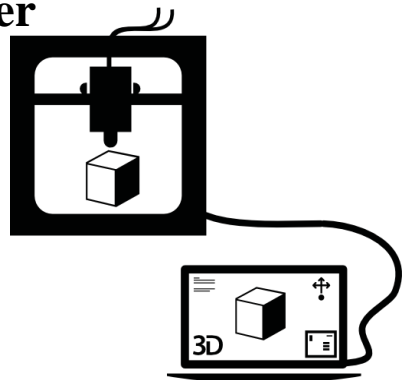# 5.3 Printing with computer

Connect Athorbot 3D

Printer to computer. And click Print in Cura after slicing.

5.4 Printing Mixed Color

5.4.1 Machine set-up

Belt is tight enough---screw is tight enough---the wire connection is correct---x/y/z axis movement is correct---auto home works fine--- temperature works fine---preheat works fine.

5.4.2 Upload Firmwares

Connect your computer and the 3D printer control box with the USB cable.
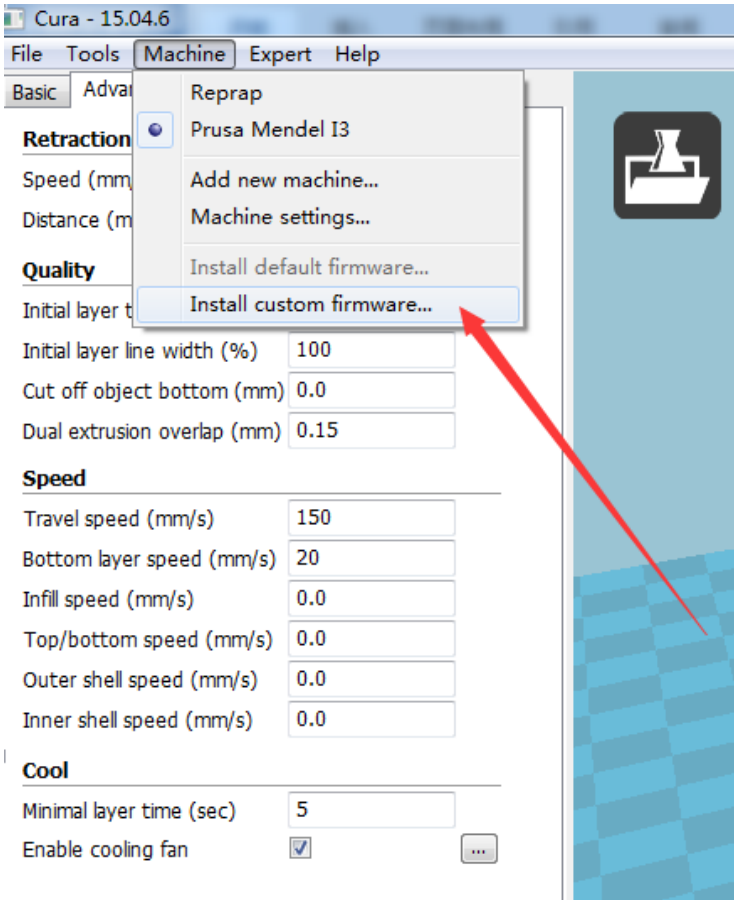
Open Cura

Load the firmware which in .hex format. (Our VIP member can download the latest firmware http://members.athorbot.com/VIP/

eg.: 3,1,1 Couple(Buddy)Mixed Color v1.0.hex
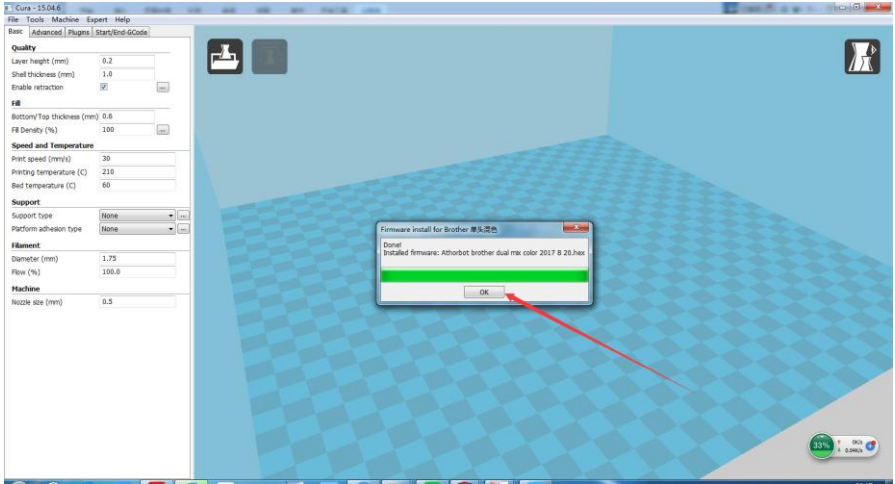
Well Done! Enjoy your printing journey!
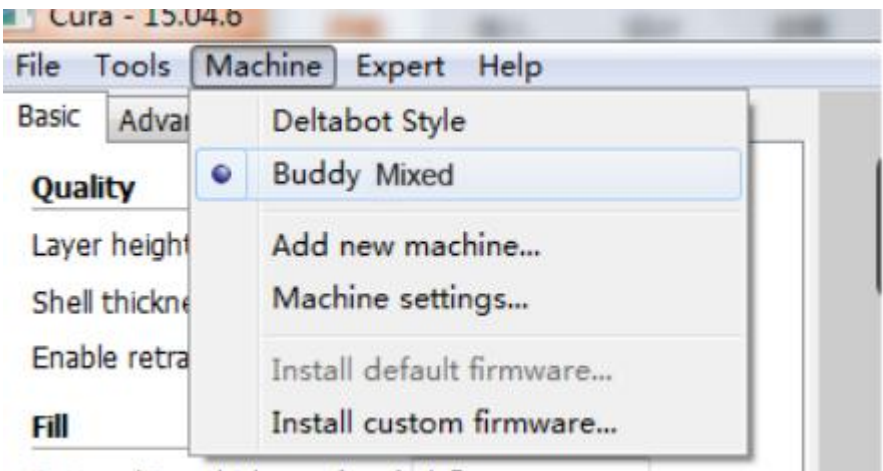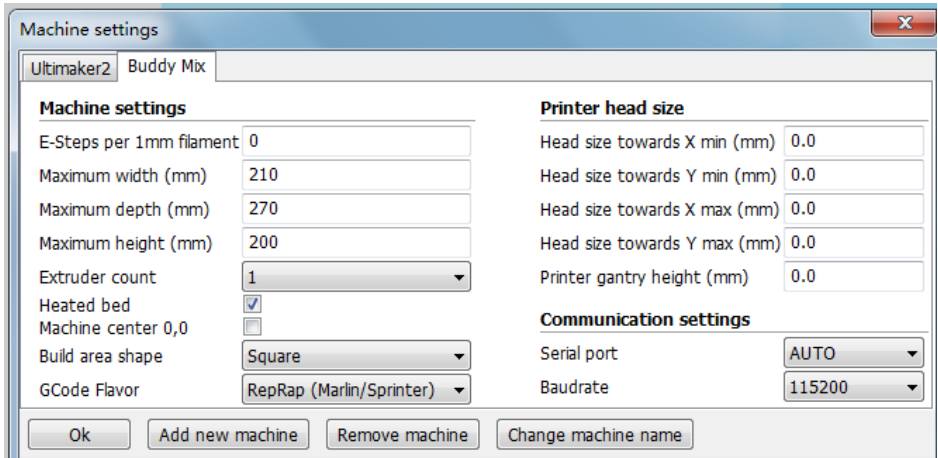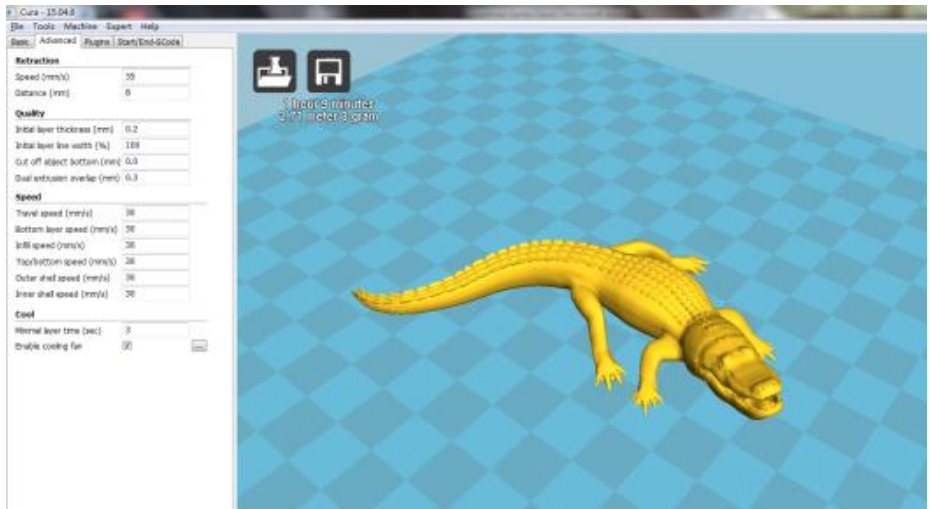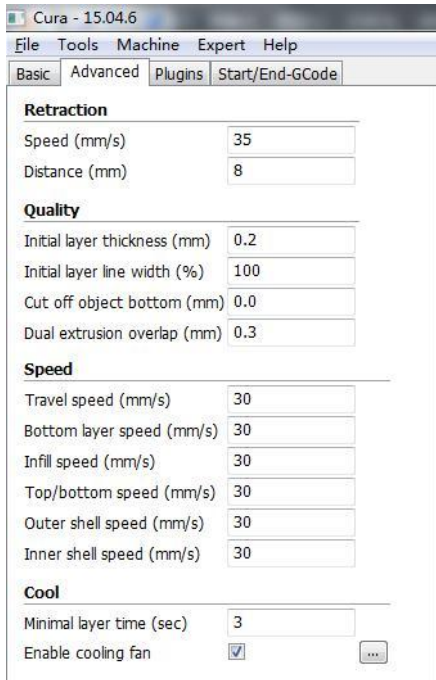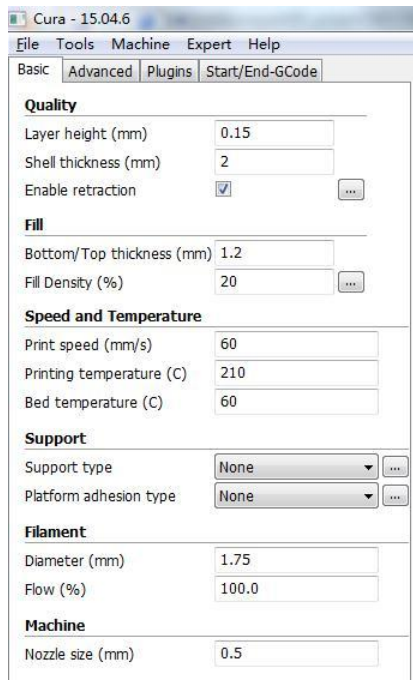
5.4.3 Slicing

Select the machine to print mixed color

Machine Parameter

Cura - 15.04.6

File Tools Machine Expert Help

Basic | Advanced | Plugins | Start/End-GCode

Quality

Layer height (mm)	0.15
Shell thickness (mm)	2
Enable retraction	☑	...

Fill

| Bottom/Top thickness (mm) | 1.2 |
| Fill Density (%) | 20 | ... |

Speed and Temperature

Print speed (mm/s)	60
Printing temperature (C)	210
Bed temperature (C)	60

Support

| Support type | None | ▼ | ... |
| Platform adhesion type | None | ▼ | ... |

Filament

| Diameter (mm) | 1.75 |
| Flow (%) | 100.0 |

Machine

| Nozzle size (mm) | 0.5 |

Cura - 15.04.6

File Tools Machine Expert Help

Basic | Advanced | Plugins | Start/End-GCode

Retraction

| Speed (mm/s) | 35 |
| Distance (mm) | 8 |

Quality

Initial layer thickness (mm)	0.2
Initial layer line width (%)	100
Cut off object bottom (mm)	0.0
Dual extrusion overlap (mm)	0.3

Speed

Travel speed (mm/s)	30
Bottom layer speed (mm/s)	30
Infill speed (mm/s)	30
Top/bottom speed (mm/s)	30
Outer shell speed (mm/s)	30
Inner shell speed (mm/s)	30

Cool

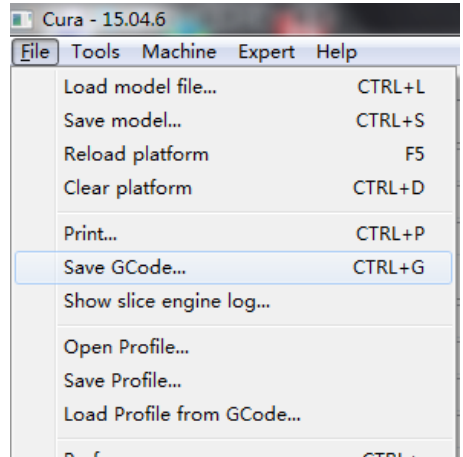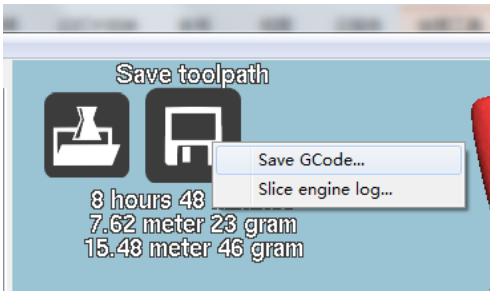| Minimal layer time (sec) | 3 |
| Enable cooling fan | ☑ | ... |

When finish slicing setting, the printing time will appear in the top left of the window. (PS: The length of printing time depends on the size of model as well as parameter setting.

8 hours 48 minutes
7.62 meter 23 gram
15.48 meter 46 gram

Saving files (Format: *Gcode)

Right click " " and choose "Save Gcode", or click "File" → "Save Gcode ".
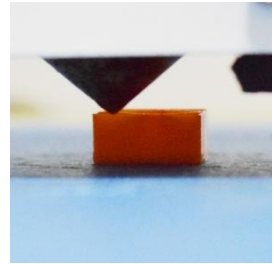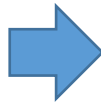
5.4.4 Change Color During Printing Process

```
Prepare          →
Control          →
>Mix_Percent     →
Print from SD    →
```
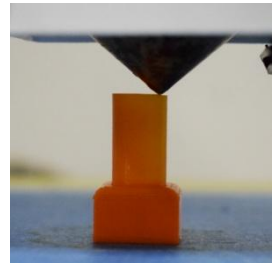
During printing, the color of prints can be change by adjusting the range of Mix-percent (Take yellow and orange filament as an example)

```
Main             ±
>MIX_0           →
 MIX_10          →
 MIX_20          →
```
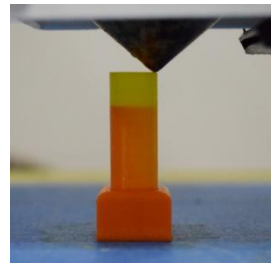
When choosing MIX_0, the color of object is orange.

```
 MIX_40          →
>MIX_50          →
 MIX_60          →
 MIX_70          →
```

When choosing MIX_50, the color of the object is mixed by yellow and orange in 1:1 ration.

```
 MIX_70          →
 MIX_80          →
 MIX_90          →
>MIX_100         →
```

When choosing MIX_100, the color of object is yellow.

Mixed Color Prints Display