



# **Slicing guide for M2P1 Hotend**

**(Base on Cura 4.7 or later)**

**V0.1**

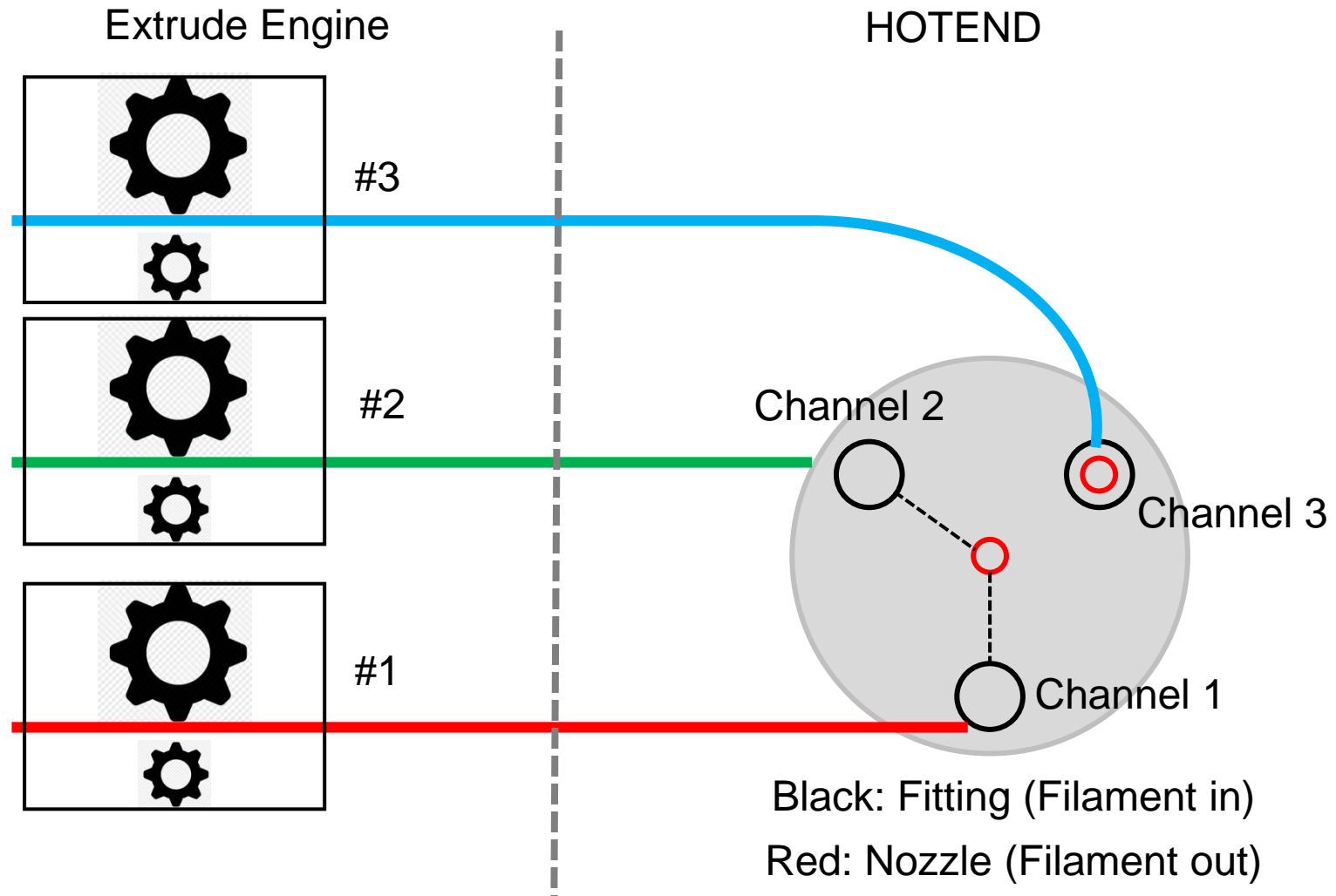
# Contents

- [Connect the extruder motors with hotend](#)
- [Download and install Cura](#)
- [Import Zonestar Printer Settings](#)
- [Setting up printer](#)
- [Setting up filament](#)
- [Parameter setting of Extruder](#)
- [Slicing](#)
- [Slicing single color 3d object](#)
- [Slicing multi colors 3d object](#)

***NOTE: For R series printers, due to the existence of multiple nozzles, the mechanical leveling step before printing is very important. Otherwise, if the nozzles are not at the same height, the nozzles will spray to the printing model during the printing process, which will lead to printing difficulties or no printing at all.***

# Connect extruder motors with hotend

There are 3 input channels and two nozzles on the M2P1 hotend, please refer to the below drawing to connect the extrude Engine with the hotend.



# Download and Install Cura

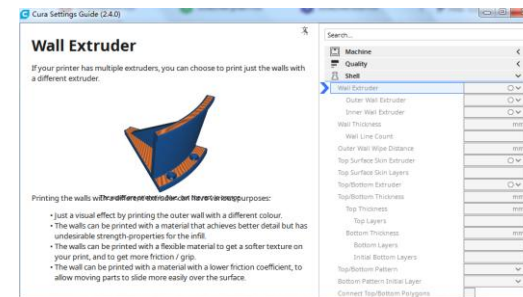
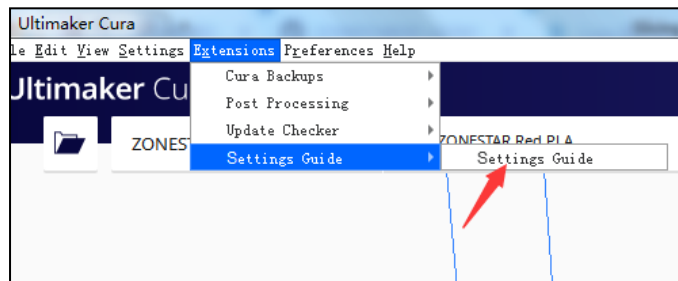
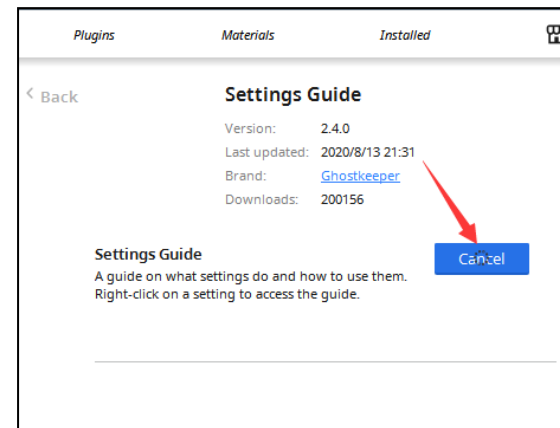
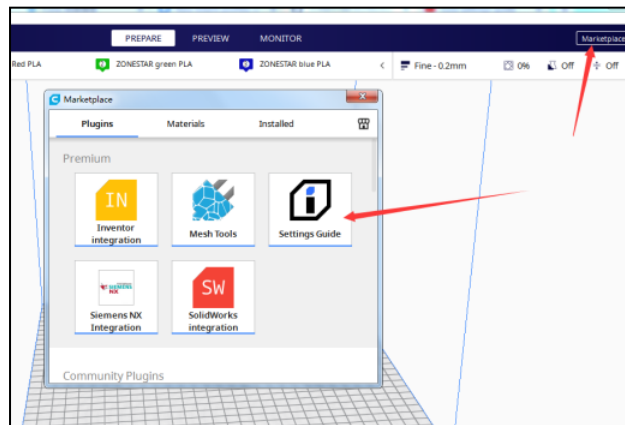
- Download cura from the below link and install it to your PC:

<https://ultimaker.com/software/ultimaker-cura>

- About how to install and use Cura, please refer to this link:

<https://support.ultimaker.com/hc/en-us/categories/360002327600>

- If you want to know more about the settings of cura, please install a “settings guide” plugin in cura, and then open it to study:

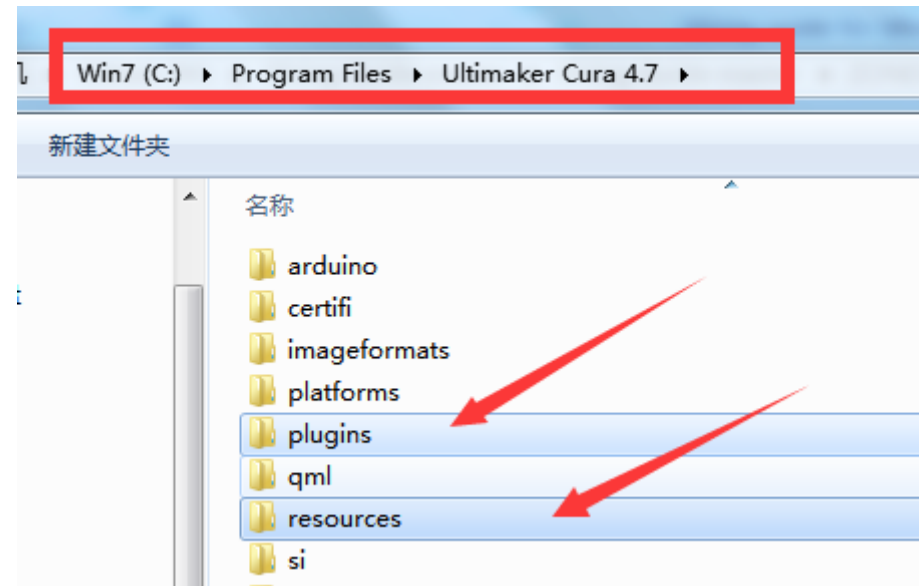
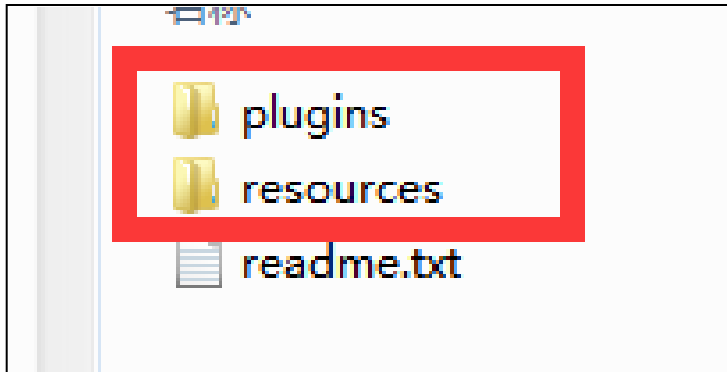


# Import ZONESTAR printer settings

1. Download "zonestar Cura Resources " from the below link:

Download link: <https://github.com/ZONESTAR3D/Document-and-User-Guide/blob/master/readme.md>

2. Copy the contents of this directory to the installation directory of Cura

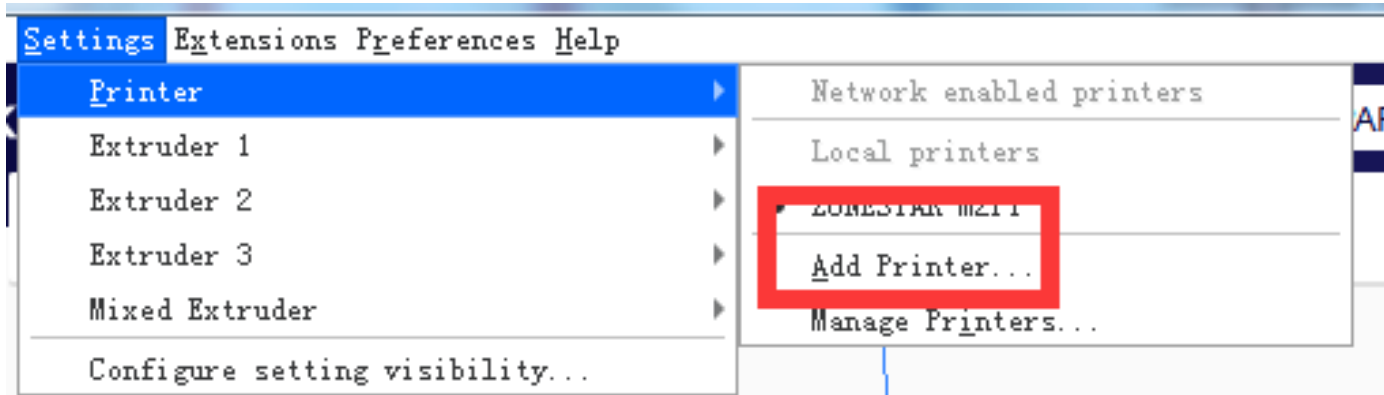


3. Run the cura

# Setting up printer

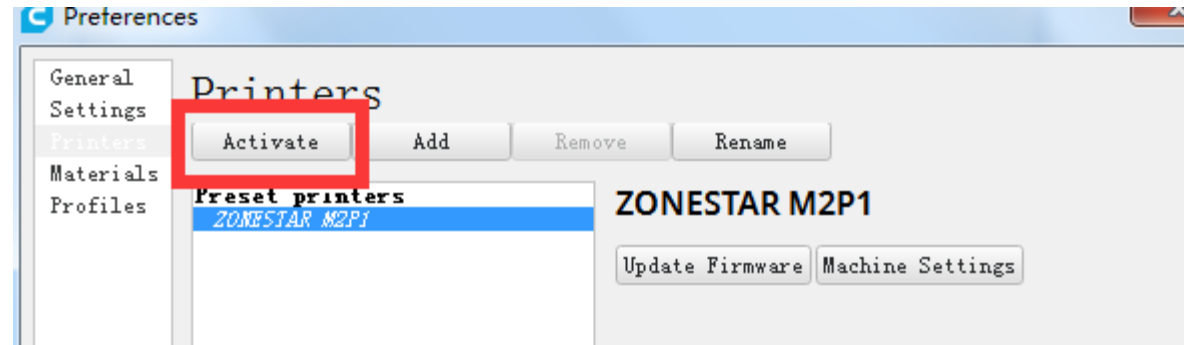
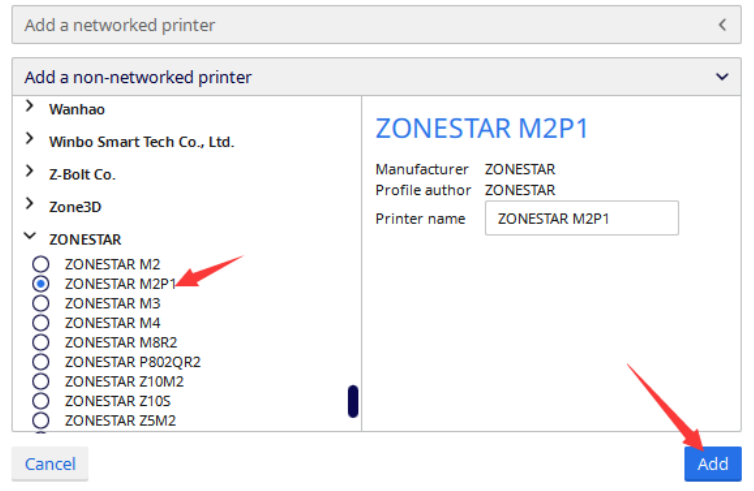
1: Open “Manage printers”

2: Click “Add”



3: Select your printer model, then click “Add”

4: Click “Activate”



# Setting up printer

Click “Machine settings”. Check the printer parameters.

Change the print size  
if your printer is  
different

**Machine Settings**

**ZONESTAR M2P1**

Printer Extruder 1 Extruder 2 Extruder 3 VisualExtr#1 VisualExtr#2 VisualExtr#3 VisualExtr#4 VisualExtr#5

**Printer Settings**

X (Width) 300 mm

Y (Depth) 300 mm

Z (Height) 400 mm

Build plate shape Rectangular

Origin at center ☐

Heated bed ☒

Heated build volume ☐

G-code flavor Marlin

**Printhead Settings**

X min -20 mm

Y min -10 mm

X max 10 mm

Y max 10 mm

Gantry Height 400 mm

Number of Extruders 8

**Start G-code**

```
G28
G1 Z15 F300
M107
;Prime the extruder
G92 E0
G1 F200 E3
```

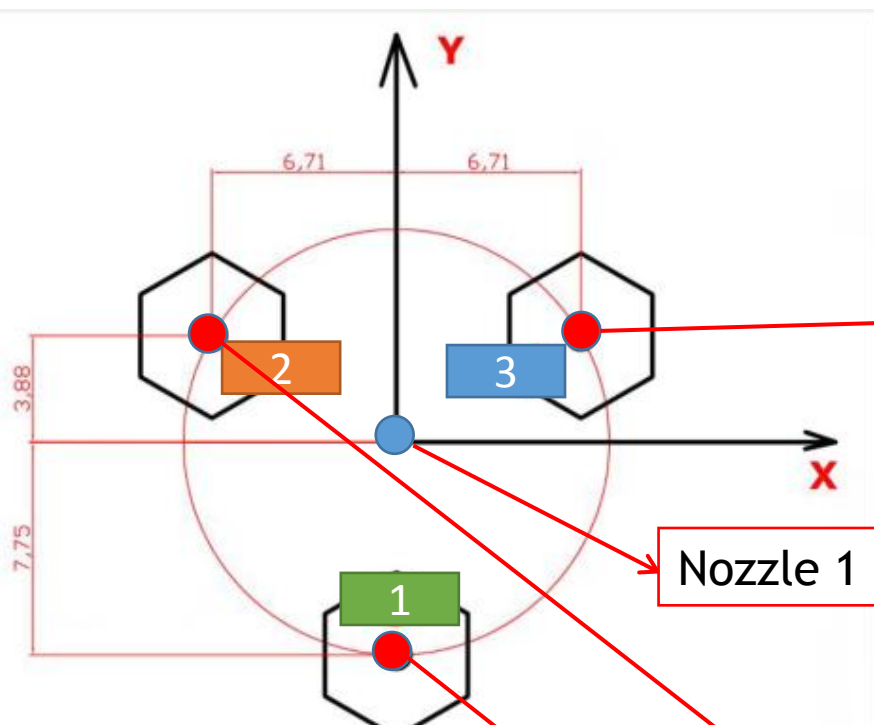
**End G-code**

```
G91
G1 E-1
G28 XY
M104 S0
G90
G92 E0
```

Close

# Set Nozzle offset

You may need to set the nozzle offset because the hotend may be rotated when installed.



Printer	Extruder 1	Extruder 2	Extruder 3	VisualExtr#1	VisualExtr#2	VisualExtr#3
<b>Nozzle Settings</b>						
Nozzle size	0.4 mm					
Compatible material diameter	1.75 mm					
Nozzle offset X	6.71 mm					
Nozzle offset Y	3.88 mm					
Cooling Fan Number	0					
<b>Extruder Start G-code</b>				<b>Extruder End G-code</b>		
<pre>;Extruder 3 start code ;feed filament into G92 E0 G1 E18 F180 G92 E0</pre>				<pre>;Extruder 3 end code ;Retract filament G92 E0 G1 E-18 F180 G92 E0</pre>		

Nozzle 1

Offset X: -6.71  
Offset Y: -3.88

If nozzle 2 at here

Offset X: 0  
Offset Y: -7.75

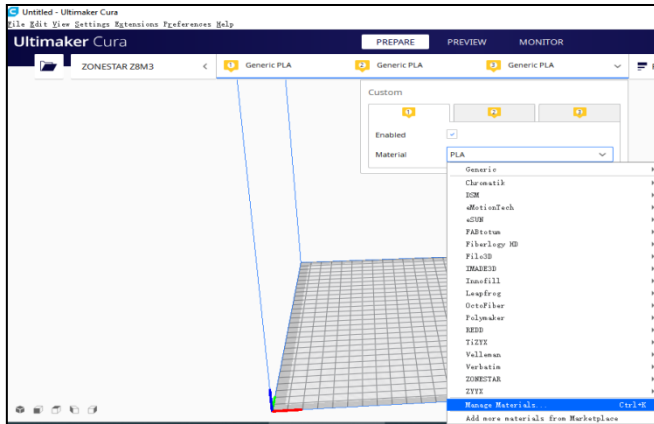
If nozzle 2 at here



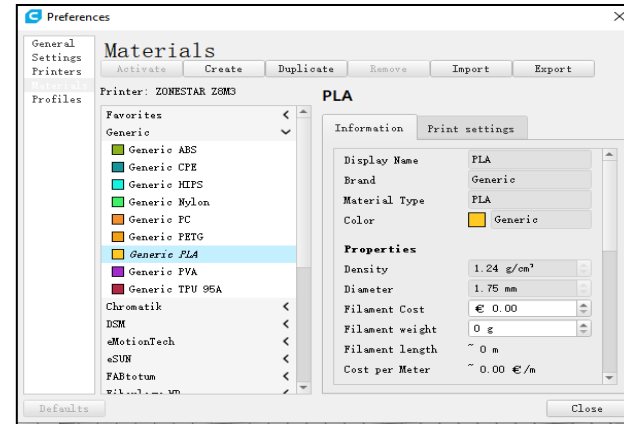
# Setting up filament

In order to easy to view when slicing, you can define the filament color

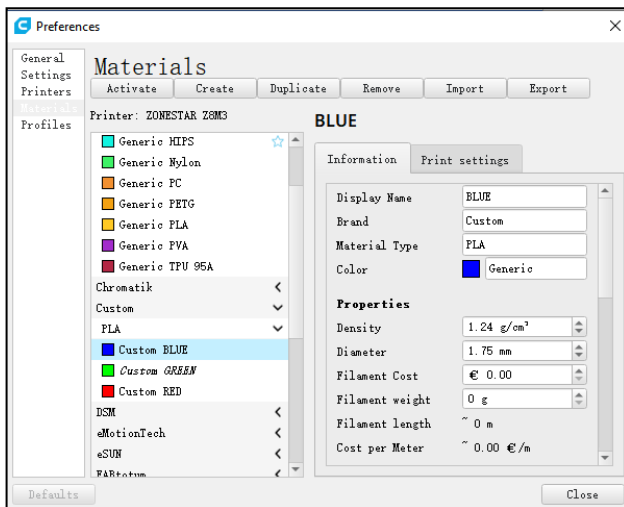
1: Open “Manage materials...”



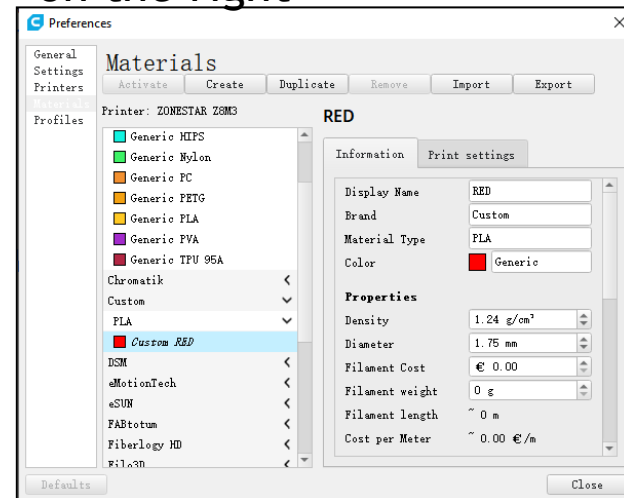
2: Click “Create”



4: Click “Duplicate”, and define more colors.

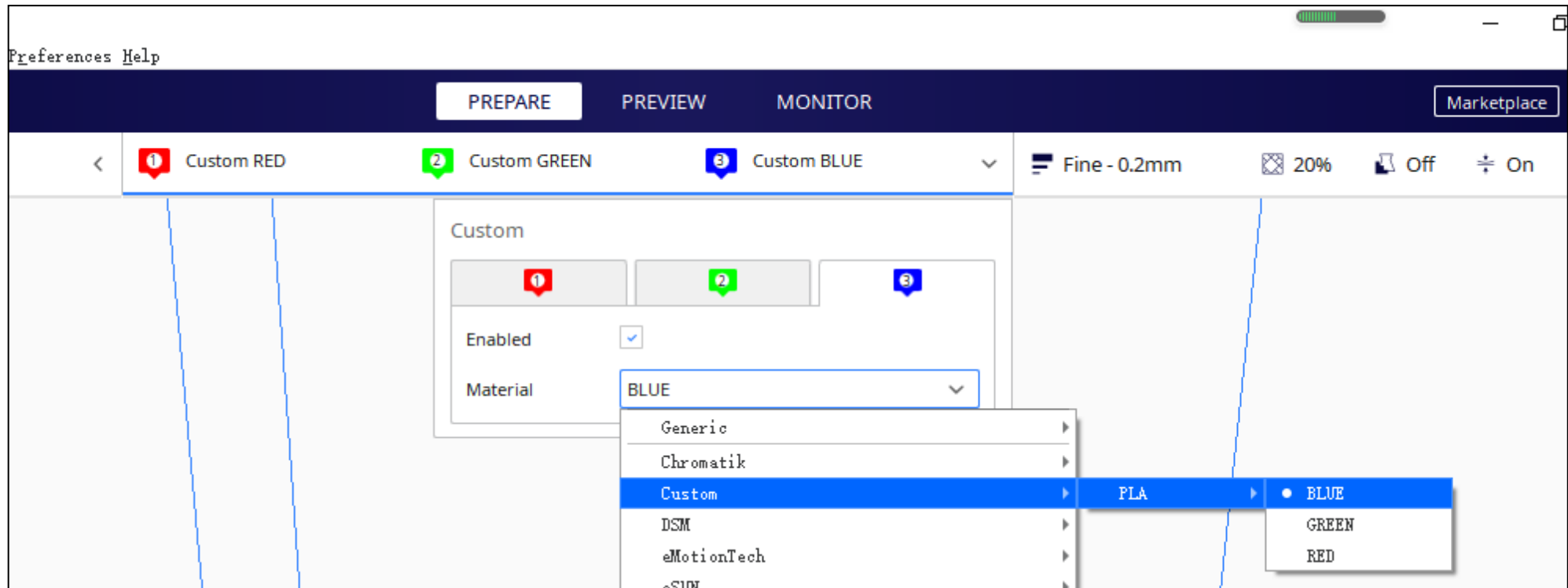


3: Find custom and modify relevant parameters in the information column on the right



# Setting up filament

Select the filament color



You can choose ZONESTAR filament too, we have set some color for PLA and PETG

# Parameter setting of Extruder

Printer Extruder 1 Extruder 2 Extruder 3

Nozzle Settings

Nozzle size 0.4 mm  
Compatible material diameter 1.75 mm  
Nozzle offset X 0 mm  
Nozzle offset Y 0 mm  
Cooling Fan Number 0

Extruder Start G-code

;Extruder 1 start code  
M163 S3 P0  
M163 S2 P0  
M163 S1 P0  
M163 S0 P100

ZONESTAR M2P1

Printer Extruder 1 Extruder 2 Extruder 3

Nozzle Settings

Nozzle size 0.4 mm  
Compatible material diameter 1.75 mm  
Nozzle offset X 0 mm  
Nozzle offset Y 0 mm  
Cooling Fan Number 0

Extruder Start G-code

;Extruder 2 start code  
M163 S3 P0  
M163 S2 P0  
M163 S1 P100  
M163 S0 P0

Printer Extruder 1 Extruder 2 Extruder 3 Extruder 4

Nozzle Settings

Nozzle size 0.4 mm  
Compatible material diameter 1.75 mm  
Nozzle offset X 6.71 mm  
Nozzle offset Y 3.88 mm  
Cooling Fan Number 0

Extruder Start G-code

;Extruder 3 start code  
M163 S3 P0  
M163 S2 P100  
M163 S1 P0  
M163 S0 P0

Printer Extruder 1 Extruder 2 Extruder 3 Visual Extruder #1

Nozzle Settings

Nozzle size 0.4 mm  
Compatible material diameter 1.75 mm  
Nozzle offset X 0 mm  
Nozzle offset Y 0 mm  
Cooling Fan Number 0

Extruder Start G-code

;Mixed Extruder #1 start code  
;NOTE:modify the mix ratio if need  
M163 S3 P0  
M163 S2 P0  
M163 S1 P50  
M163 S0 P50  
M164 S3  
T3

In addition to the actual extruder, five virtual extruders are set in the machine settings. Virtual extruder refers to mixing the filament of extruder 1 and 2 to get a new color filament.

You can modify the mix rate if need, for example:

M163 S3 P0  
M163 S2 P0  
M163 S1 P10  
M163 S0 P90

; M163 command set the ratio of extruder

M164 S3

;M164 command set these ratio to the Extruder 3

T3

;switch the current Extruder #3 (Apply the mix ratio to extrude motor)

# Slicing

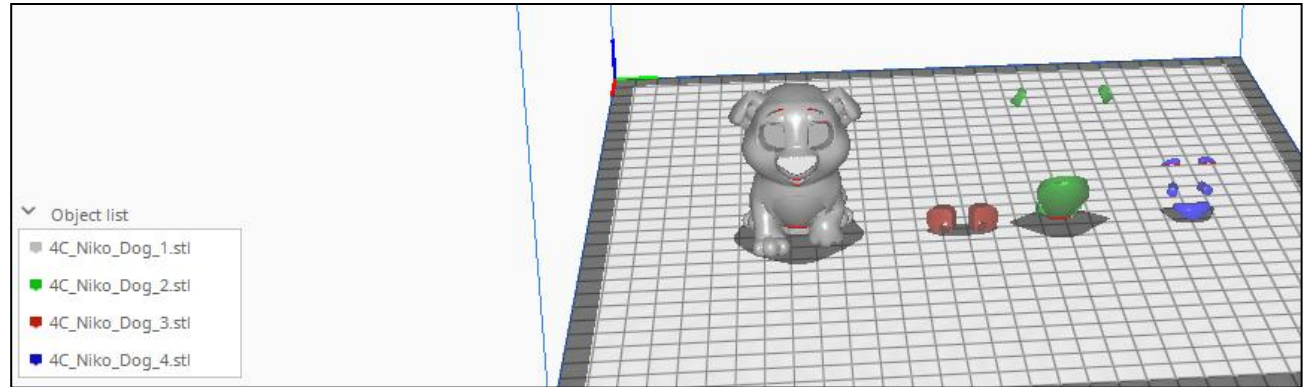
As an example, we will use the same 3d object in the following pages. This 3d object is a 4-color model, which has already divided into four parts

 4C\_Niko\_Dog\_1.stl

 4C\_Niko\_Dog\_2.stl

 4C\_Niko\_Dog\_3.stl

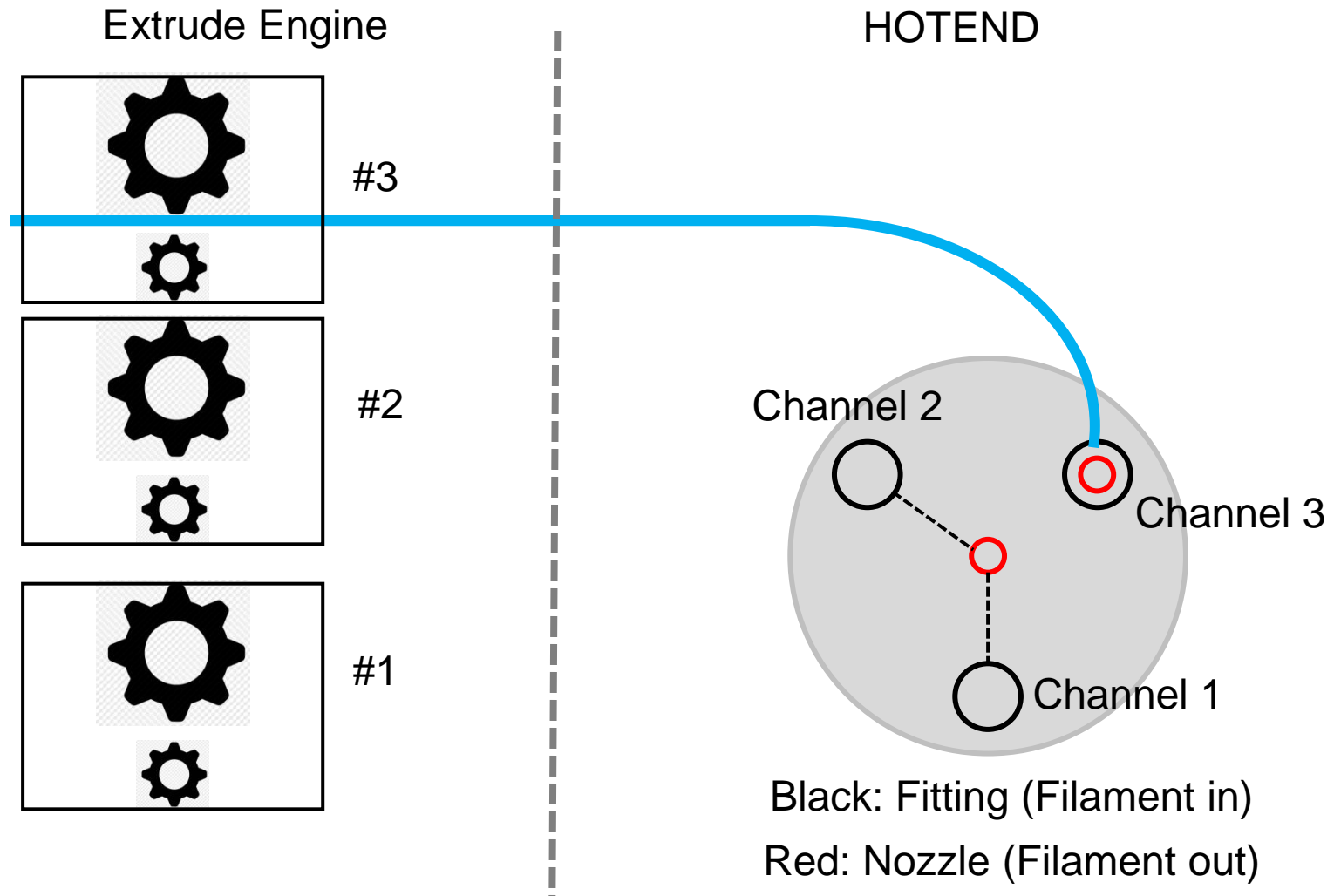
 4C\_Niko\_Dog\_4.stl



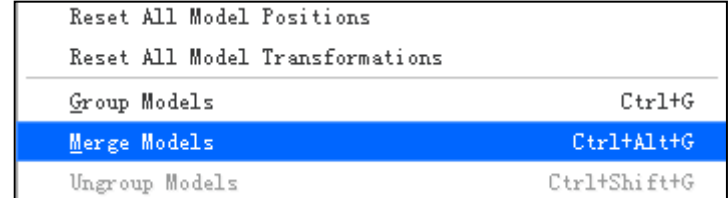
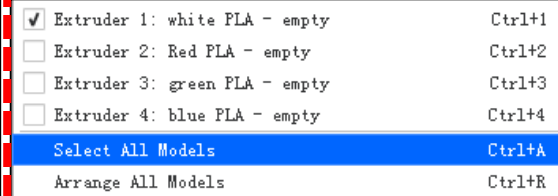
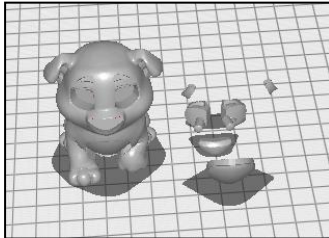
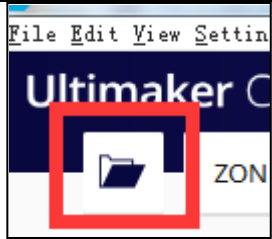
**NOTE:** If you need to print multicolor, you need a 3d object that has been divided (the number of divided parts is according to the number of colors), and their origin position must be consistent in order to be merged.

# Load filament for printing 1 color 3d object

Load filament from extruder engine #3 and load filament into that independent channel



# Slicing steps for 1 color 3d object

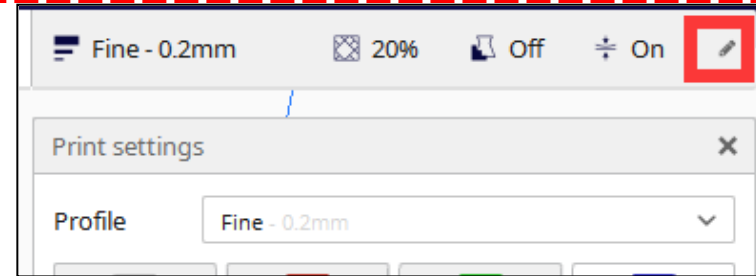
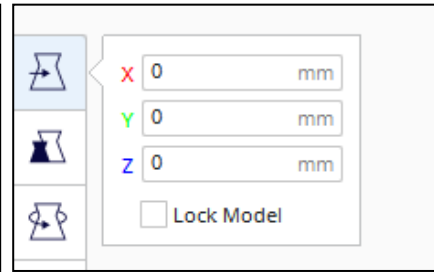
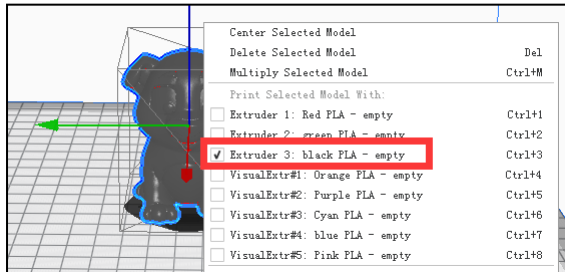


Load files

Right click the mouse  
Select all models

Merge

Pass these two steps if the stl file is one color



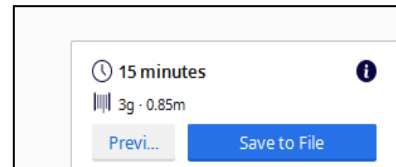
Assign extruder 3  
to print (1)

Move/Scale/Rotate  
the model

Set slicing parameter  
(2)

NOTE1: use a separate  
channel to print single color

NOTE2: load "ZMZ2P1\_1C\_Niko\_Dog.3mf" to  
see the slicing settings



Slicing it

save it

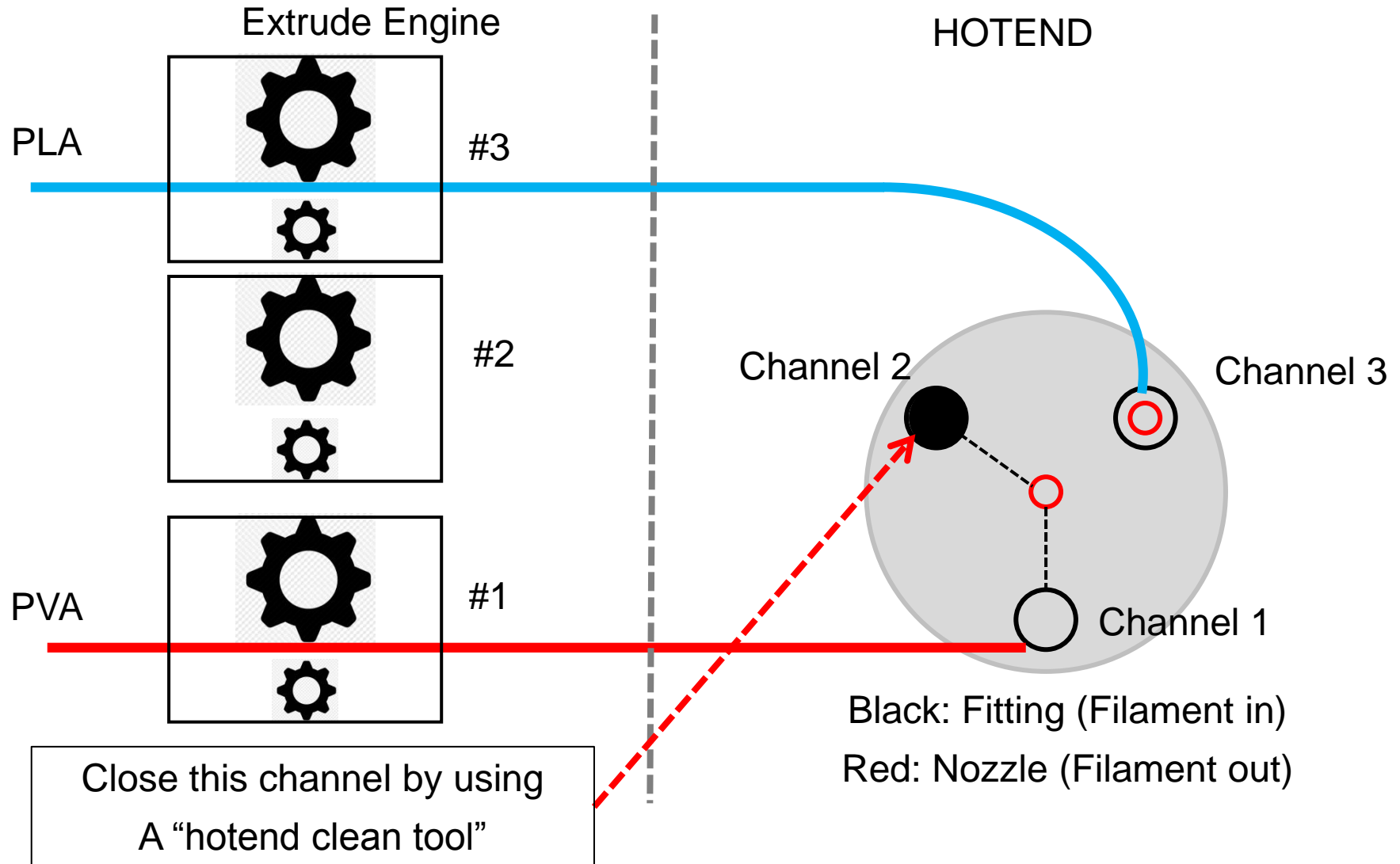
Copy the gcode file to SD card,  
then print it

# Settings for Slicing 1 color 3d object

Quality			Infill			Travel		
Layer Height	<input type="text" value="0.2"/>	mm	Infill Extruder	<input type="text" value="Extruder 3"/>		Enable Retraction	<input checked="" type="checkbox"/>	
Initial Layer Height	<input type="text" value="0.3"/>	mm	Infill Density	<input type="text" value="20"/>	%	Retract at Layer Change	<input checked="" type="checkbox"/>	
Line Width	<input type="text" value="0.4"/>	mm	Infill Line Distance	<input type="text" value="4.0"/>	mm	Retraction Distance	<input type="text" value="10"/>	mm
Wall Line Width	<input type="text" value="0.4"/>	mm	Infill Pattern	<input type="text" value="Grid"/>		Retraction Speed	<input type="text" value="25"/>	mm/s
Outer Wall Line Width	<input type="text" value="0.4"/>	mm	Infill Line Multiplier	<input type="text" value="1"/>		Combing Mode	<input type="text" value="All"/>	
Inner Wall(s) Line Width	<input type="text" value="0.4"/>	mm	Infill Overlap Percentage	<input type="text" value="10"/>	%	Avoid Printed Parts When Traveling	<input checked="" type="checkbox"/>	
Top/Bottom Line Width	<input type="text" value="0.4"/>	mm	Infill Layer Thickness	<input type="text" value="0.2"/>	mm	Avoid Supports When Traveling	<input type="checkbox"/>	
Infill Line Width	<input type="text" value="0.4"/>	mm	Gradual Infill Steps	<input type="text" value="0"/>		Travel Avoid Distance	<input type="text" value="0.625"/>	mm
Initial Layer Line Width	<input type="text" value="100.0"/>	%				Z Hop When Retracted	<input checked="" type="checkbox"/>	
						Z Hop Only Over Printed Parts	<input type="checkbox"/>	
						Z Hop Height	<input type="text" value="1"/>	mm
						Z Hop After Extruder Switch	<input type="checkbox"/>	
Shell			Material			Cooling		
Wall Extruder	<input type="text" value="Extruder 3"/>		Printing Temperature	<input type="text" value="195"/>	°C	Enable Print Cooling	<input checked="" type="checkbox"/>	
Outer Wall Extruder	<input type="text" value="Extruder 3"/>		Printing Temperature Initial Layer	<input type="text" value="195"/>	°C	Fan Speed	<input type="text" value="100.0"/>	%
Inner Wall Extruder	<input type="text" value="Extruder 3"/>		Initial Printing Temperature	<input type="text" value="195"/>	°C	Regular Fan Speed	<input type="text" value="100.0"/>	%
Wall Thickness	<input type="text" value="0.8"/>	mm	Final Printing Temperature	<input type="text" value="195"/>	°C	Maximum Fan Speed	<input type="text" value="100.0"/>	%
Wall Line Count	<input type="text" value="2"/>		Build Plate Temperature	<input type="text" value="60"/>	°C	Regular/Maximum ... Speed Threshold	<input type="text" value="10"/>	s
Top/Bottom Extruder	<input type="text" value="Not overri..."/>		Build Plate Temperature Initial Layer	<input type="text" value="60"/>	°C	Initial Fan Speed	<input type="text" value="0"/>	%
Top/Bottom Thickness	<input type="text" value="0.8"/>	mm	Standby Temperature	<input type="text" value="195"/>	°C	Regular Fan Speed at Height	<input type="text" value="0.3"/>	mm
Top Thickness	<input type="text" value="0.8"/>	mm				Regular Fan Speed at Layer	<input type="text" value="2"/>	
Top Layers	<input type="text" value="4"/>		Speed			Minimum Layer Time	<input type="text" value="5"/>	s
Bottom Thickness	<input type="text" value="0.8"/>	mm	Print Speed	<input type="text" value="50"/>	mm/s	Minimum Speed	<input type="text" value="10"/>	mm/s
Bottom Layers	<input type="text" value="4"/>		Infill Speed	<input type="text" value="50"/>	mm/s	Lift Head	<input type="checkbox"/>	
Optimize Wall Printing Order	<input checked="" type="checkbox"/>		Wall Speed	<input type="text" value="25.0"/>	mm/s			
Fill Gaps Between Walls	<input type="text" value="Everywhere"/>		Outer Wall Speed	<input type="text" value="25.0"/>	mm/s	Build Plate Adhesion		
Horizontal Expansion	<input type="text" value="0"/>	mm	Inner Wall Speed	<input type="text" value="50.0"/>	mm/s	Build Plate Adhesion Type	<input type="text" value="Skirt"/>	
Enable Ironing	<input type="checkbox"/>		Top/Bottom Speed	<input type="text" value="25.0"/>	mm/s	Build Plate Adhesion Extruder	<input type="text" value="Extruder 3"/>	
			Travel Speed	<input type="text" value="100"/>	mm/s	Skirt Line Count	<input type="text" value="3"/>	
			Initial Layer Speed	<input type="text" value="25.0"/>	mm/s			
			Skirt/Brim Speed	<input type="text" value="25.0"/>	mm/s			
			Enable Acceleration Control	<input checked="" type="checkbox"/>				
			Enable Jerk Control	<input checked="" type="checkbox"/>				

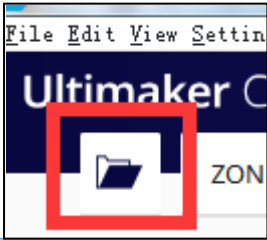
# Use water-soluble materials

Some models need support to print well, you can load main filament (PLA/ABS) in to channel #3 and support filament (PVA/HIPS) into Channel #1, and closed Channel #2.

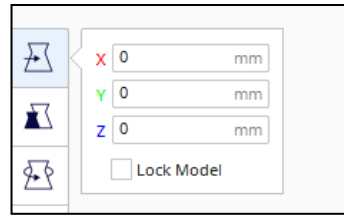




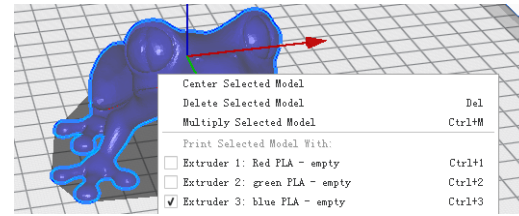
# Slicing steps for 1 color 3d object + support



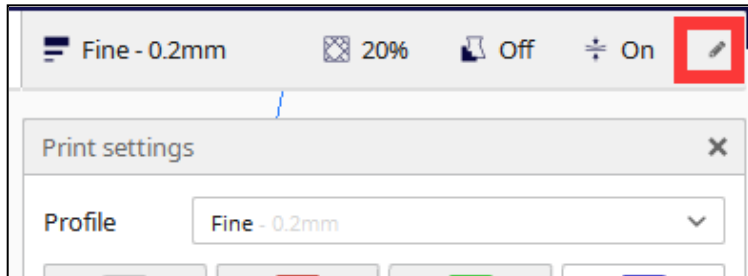
Load files



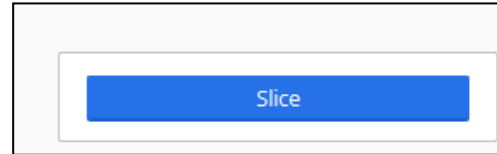
Move/Scale/Rotate  
the model



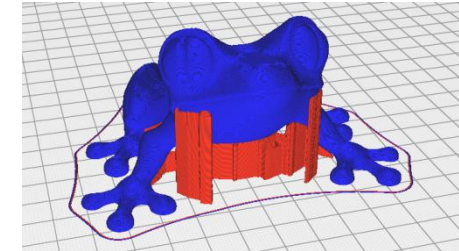
Assign to extruder 3



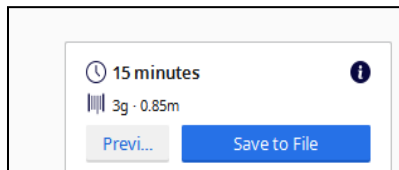
Set slicing parameter  
(2)



Slicing it



Pre-View



save it

Example: ZM2P1\_frog

Copy the gcode file to SD card and  
print it

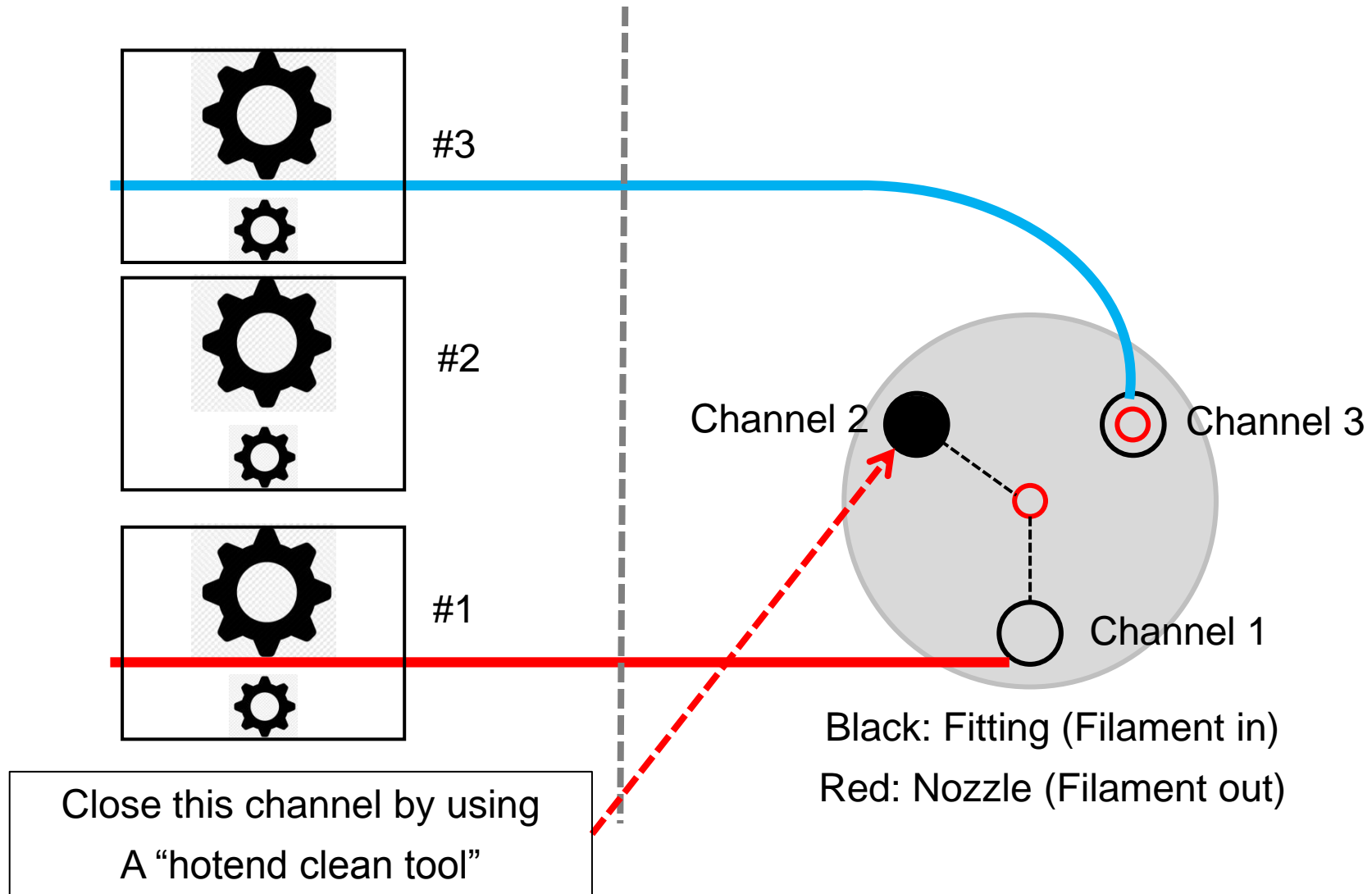
# Settings for Slicing 1 color 3d object

Material	
Printing Temperature	195 °C
Printing Temperature Initial Layer	195 °C
Initial Printing Temperature	195 °C
Final Printing Temperature	195 °C
Build Plate Temperature	60 °C
Build Plate Temperature Initial Layer	60 °C
Standby Temperature	195 °C
Generate Support <input checked="" type="checkbox"/>	
Support Extruder	Extruder 1
Support Infill Extruder	Extruder 1
First Layer Support Extruder	Extruder 1
Support Interface Extruder	Extruder 1
Support Roof Extruder	Extruder 1
Support Floor Extruder	Extruder 1
Support Structure	Normal
Support Placement	Touching Buil...
Support Overhang Angle	50 °
Support Pattern	Grid

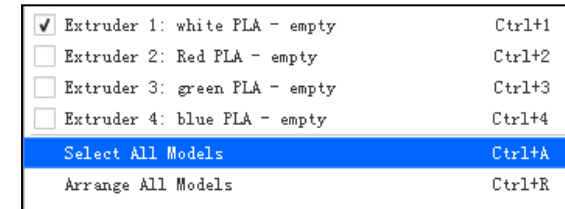
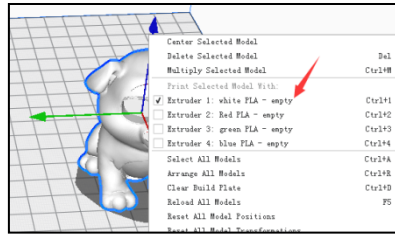
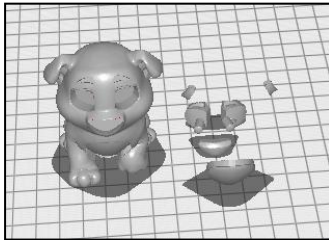
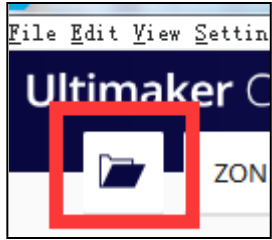
Support Wall Line Count	1
Connect Support Lines	<input checked="" type="checkbox"/>
Support Density	15 %
Support Line Distance	5.3333 mm
Initial Layer Support Line Distance	5.3333 mm
Support Z Distance	0 mm
Support Top Distance	0 mm
Support X/Y Distance	0.7 mm
Support Distance Priority	Z overrides X/Y
Minimum Support X/Y Distance	0.2 mm
Support Stair Step Height	0.3 mm
Support Stair Step Maximum Width	5.0 mm
Support Join Distance	2.0 mm
Support Horizontal Expansion	0 mm
Support Infill Layer Thickness	0.2 mm
Gradual Support Infill Steps	0
Minimum Support Area	0.0 mm <sup>2</sup>
Enable Support Interface	<input type="checkbox"/>
Enable Support Roof	<input type="checkbox"/>
Enable Support Floor	<input type="checkbox"/>
Use Towers	<input checked="" type="checkbox"/>
Tower Diameter	3.0 mm
Maximum Tower-Sorted Diameter	3.0 mm
Tower Roof Angle	65 °

# Load filament for 2 colors 3d object - mode 1

You can use two independent nozzles to print two colors, and use oozing shield to deal with oozing issue



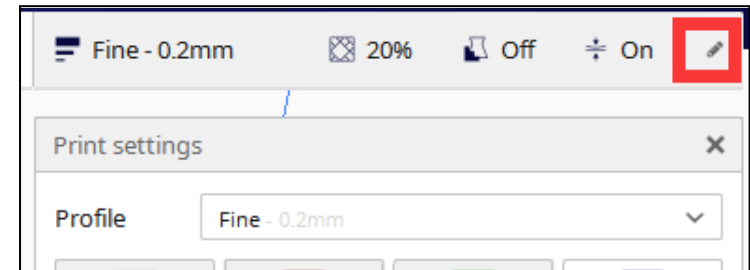
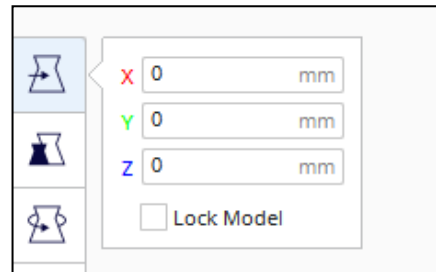
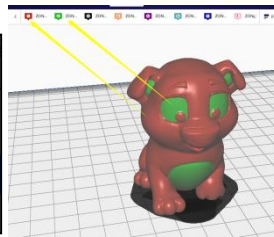
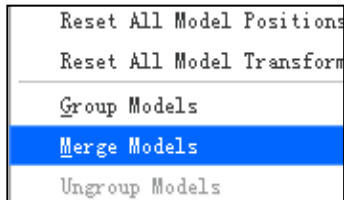
# Slicing 2 colors 3d object (model 1)



Load files

Right click the part and  
assign extruder for each

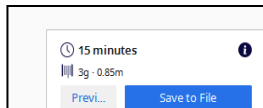
Right click the mouse  
Select all models



Merge

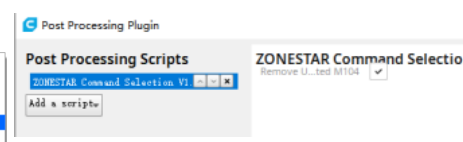
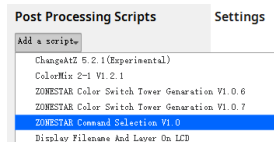
Move/Scale/Rotate  
the model

Set slicing parameter



ZZM2P1\_2CM1\_Niko\_Dog\_Orig.gcode

slicing and save it to PC



Apply  
Command Selection Plugin

ZZM2P1\_2CM1\_Niko\_Dog.gcode

Copy to SD card  
and print it

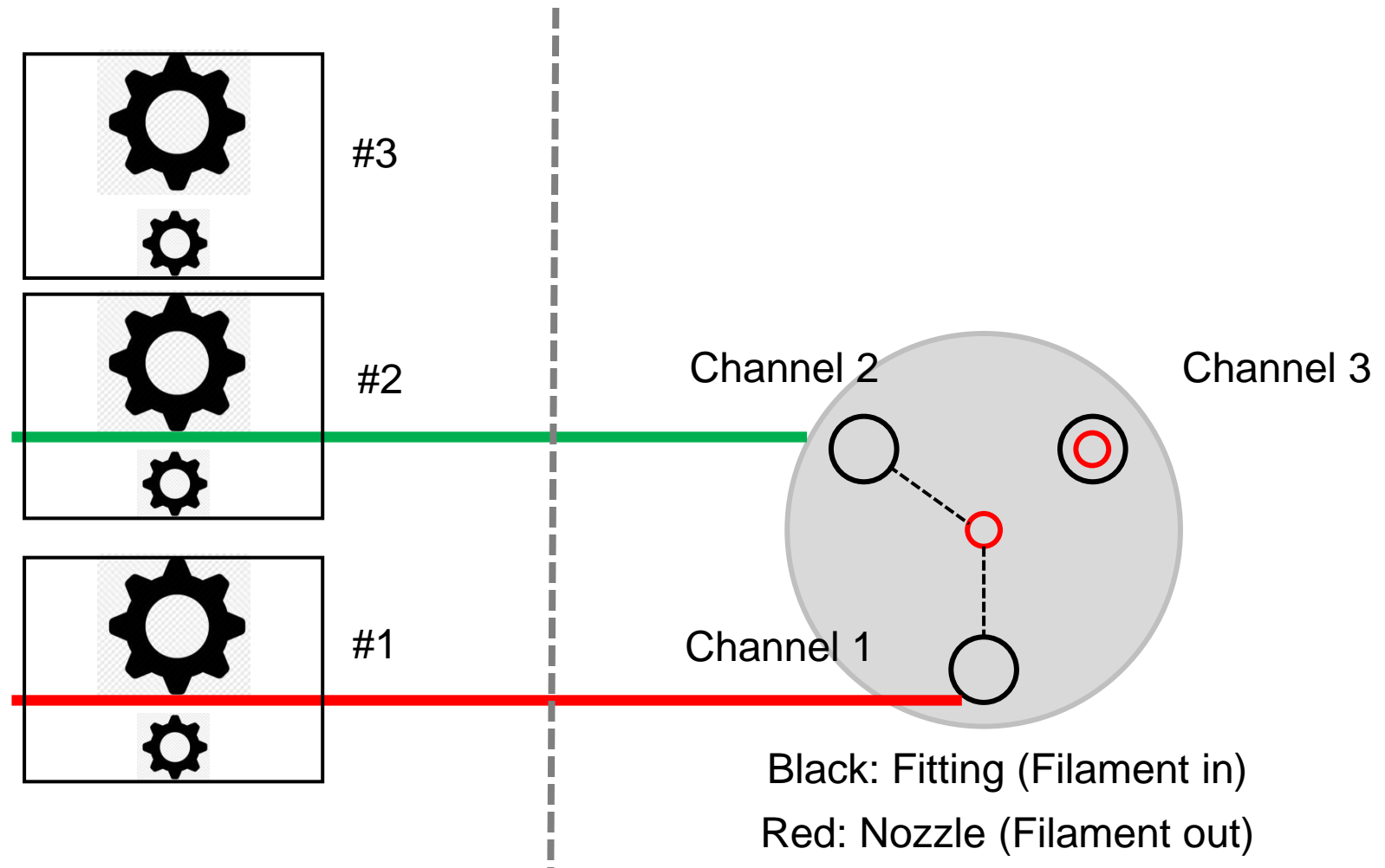
# Settings for Slicing 2 colors 3d object

Material		<
Printing Temperature	195	°C
Printing Temperature Initial Layer	195	°C
Initial Printing Temperature	195	°C
Final Printing Temperature	195	°C
Build Plate Temperature	60	°C
Build Plate Temperature Initial Layer	60	°C
Flow	100	%
Wall Flow	100	%
Outer Wall Flow	100	%
Inner Wall(s) Flow	100	%
Top/Bottom Flow	100	%
Infill Flow	100	%
Standby Temperature	195	°C

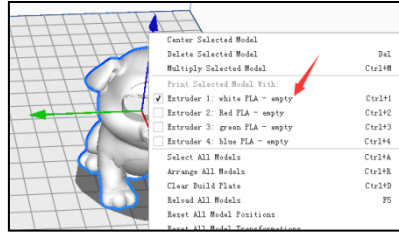
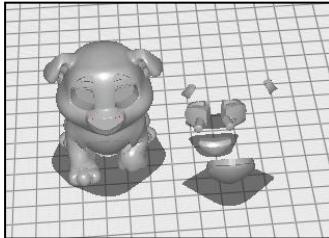
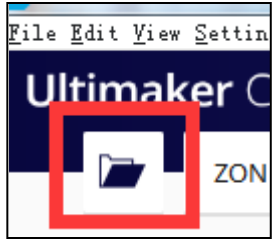
Build Plate Adhesion		>
Dual Extrusion		
Enable Prime Tower	<input type="checkbox"/>	
Enable Ooze Shield	<input checked="" type="checkbox"/>	
Ooze Shield Angle	60	°
Ooze Shield Distance	2	mm
Nozzle Switch Retraction Distance	10	mm
Nozzle Switch Retraction Speed	25	mm/s
Nozzle Switch Retract Speed	25	mm/s
Nozzle Switch Prime Speed	25	mm/s

# Load filament for 2 colors 3d object - mode 2

You can also use the mixing color channels to print two color 3d object, there is only one nozzle so that there will be no oozing issue, but you need to set a color switch tower, it will take more time and filament.



# Slicing 2 colors 3d object - model 2

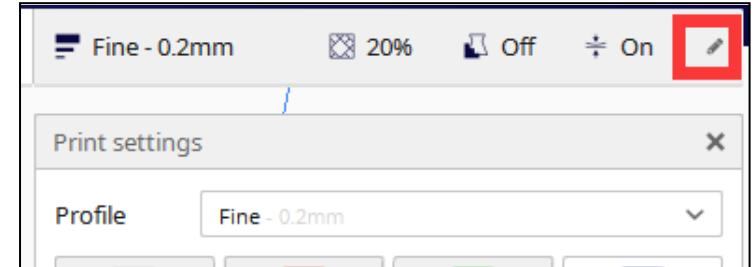
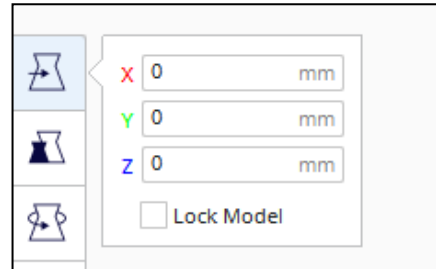
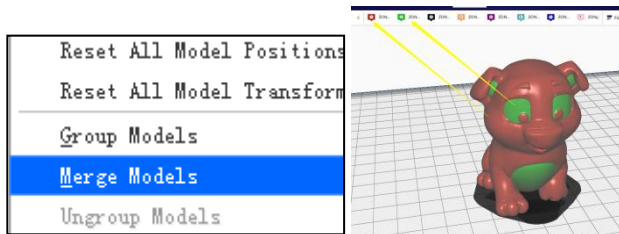


<input checked="" type="checkbox"/>	Extruder 1: white PLA - empty	Ctrl+1
<input type="checkbox"/>	Extruder 2: Red PLA - empty	Ctrl+2
<input type="checkbox"/>	Extruder 3: green PLA - empty	Ctrl+3
<input type="checkbox"/>	Extruder 4: blue PLA - empty	Ctrl+4
<input checked="" type="checkbox"/>	Select All Models	Ctrl+A
<input type="checkbox"/>	Arrange All Models	Ctrl+R

Load files

Right click the part and  
assign extruder for each

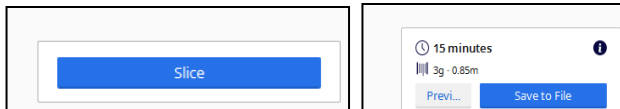
Right click the mouse  
Setlect all models



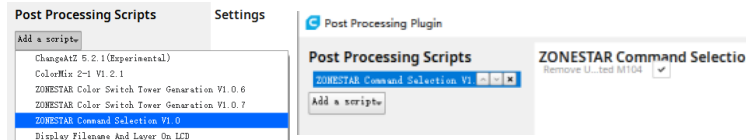
Merge

Move/Scale/Rotate  
the model

Set slicing parameter



ZZM2P1\_2CM2\_Niko\_Dog\_Orig.gcode



ZZM2P1\_2CM2\_Niko\_Dog.gcode

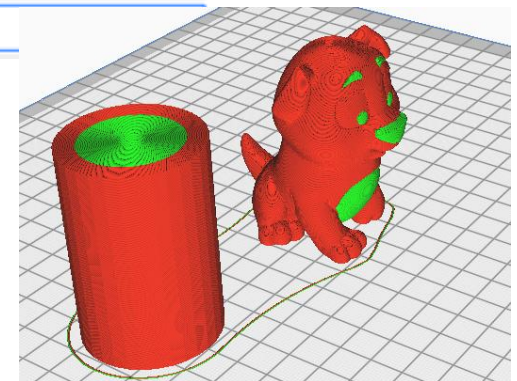
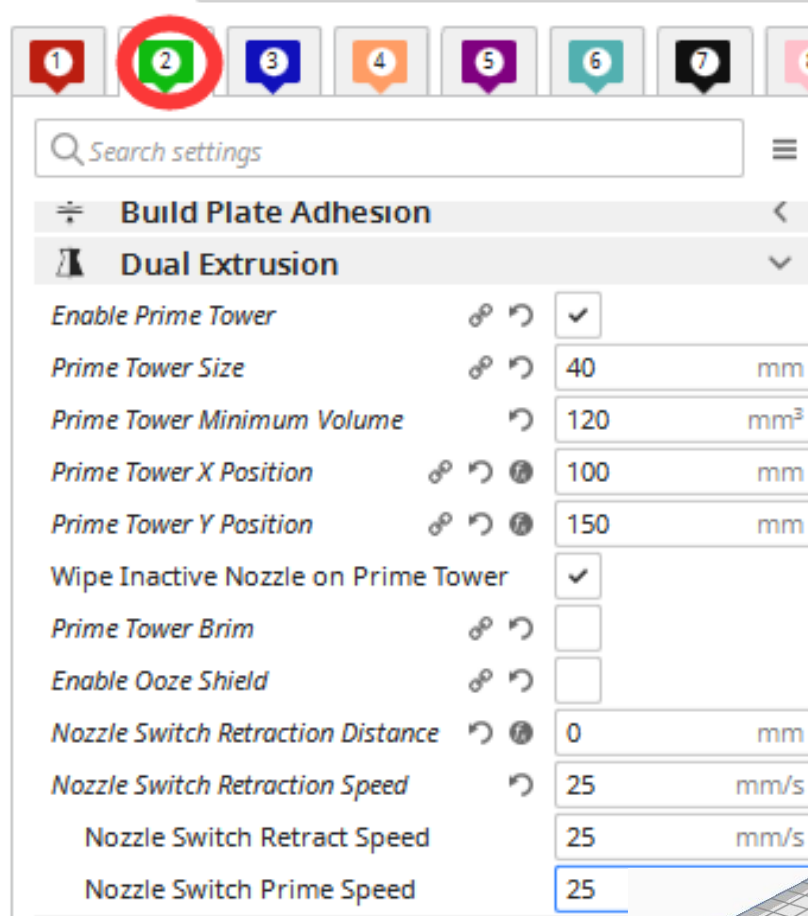
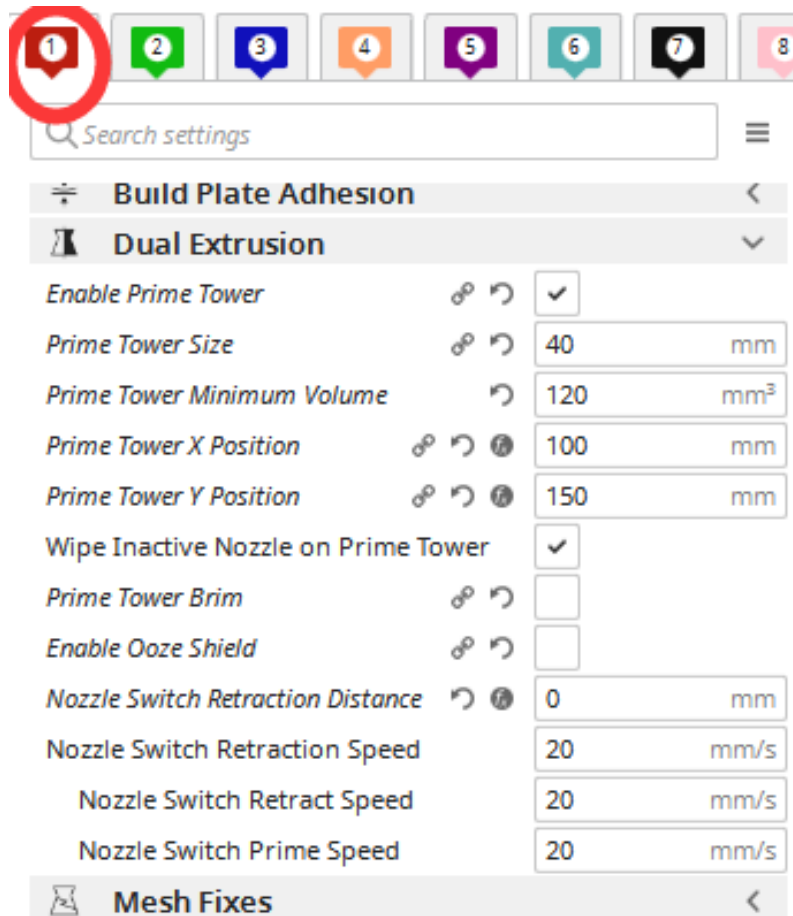
slicing and save it to PC

Apply  
Command Selection Plugin

Copy to SD card  
and print it

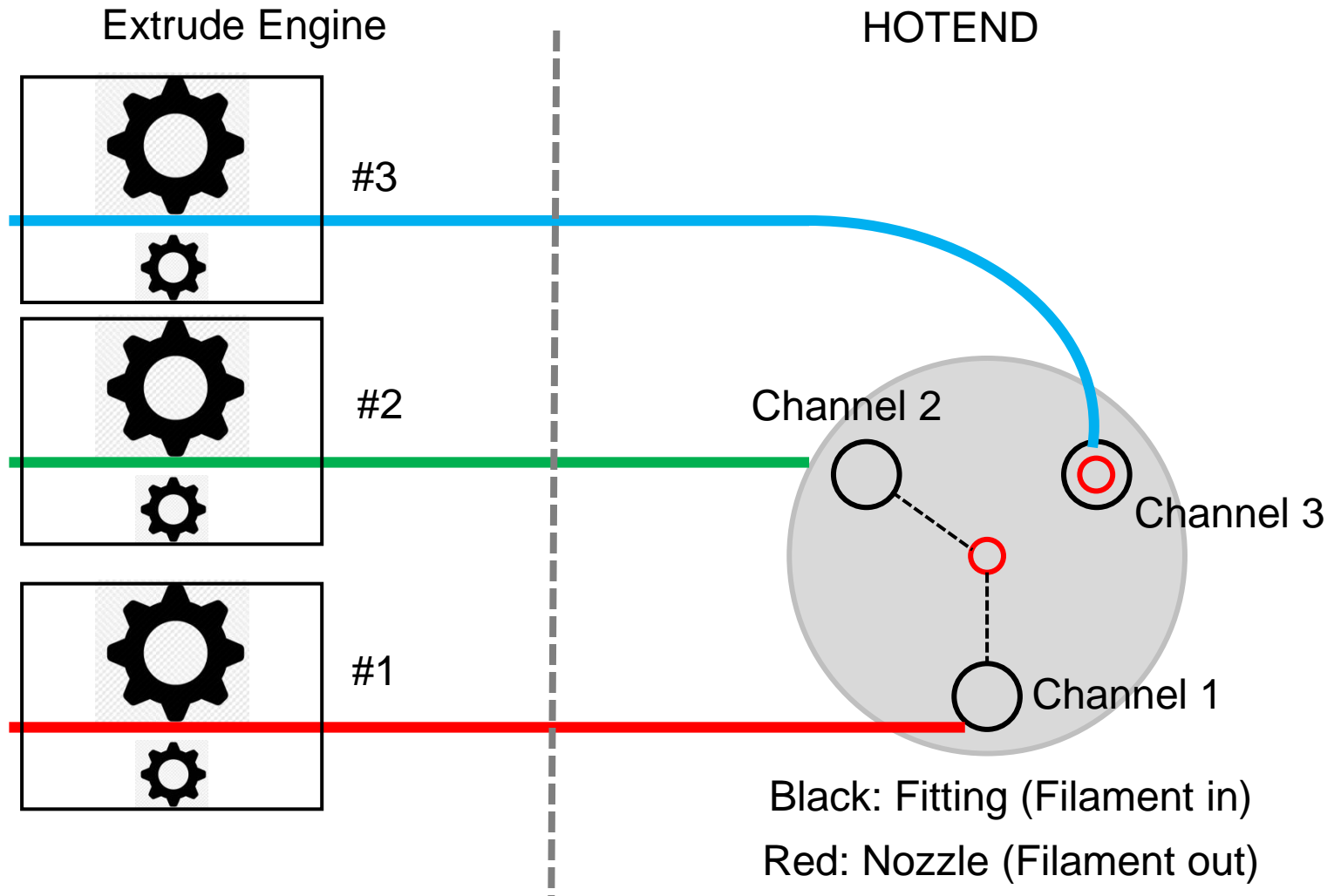


# Settings for Slicing 2 colors 3d object - mode 2





# Load filament for 3~8 colors 3d object



# Slicing 3 colors 3d object

# Slicing 4~8 colors 3d object