Zonestar

Slicing guide for Multi Color Printer

(Base on Cura 4.7 or later)

V1.0

Contents

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.

Download and install Cura

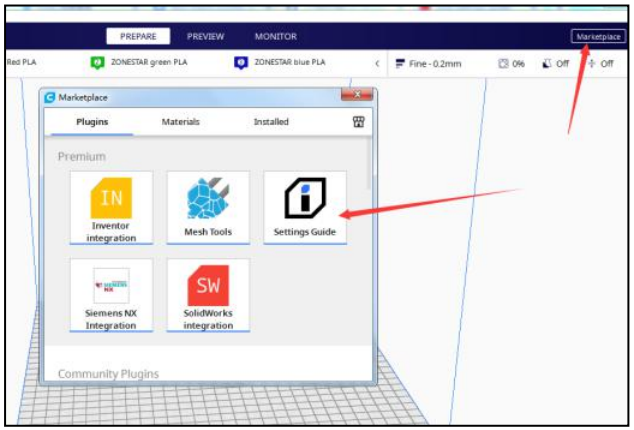
- Download cura from the below link and install it to your PC:

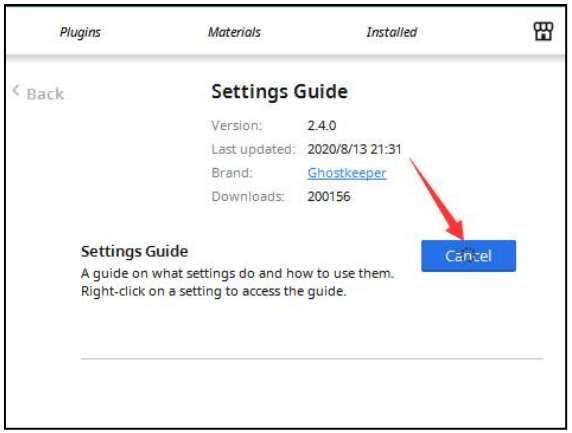
https://ultimaker.com/software/ultimaker-cura

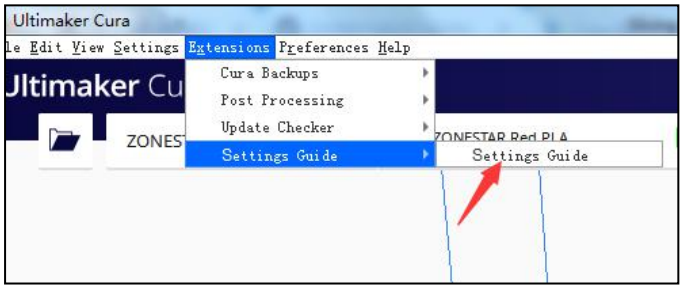
- About how to install and use Cura, please refer to this link:

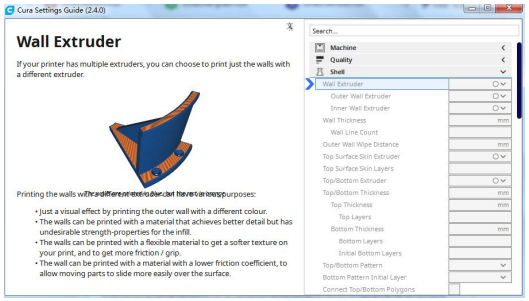
https://support.ultimaker.com/hc/en-us/categories/360002327600

- 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:







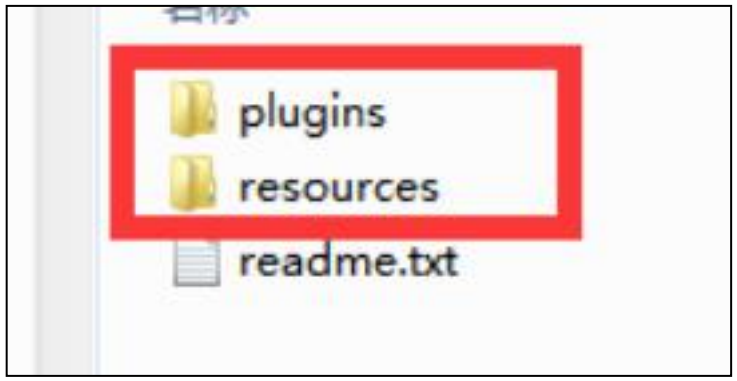


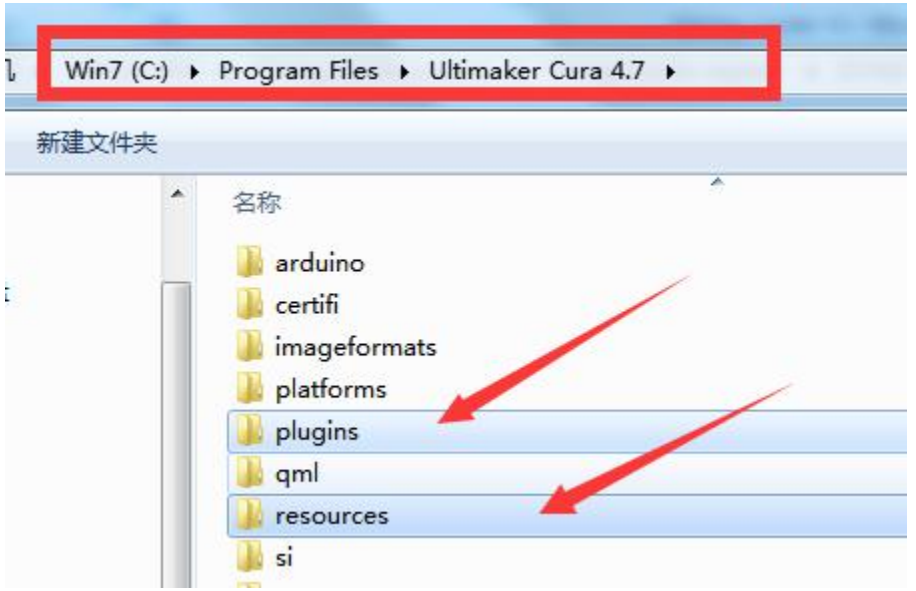
Import ZONESTAR printer settings

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 sirctory to the installation directory of Cura

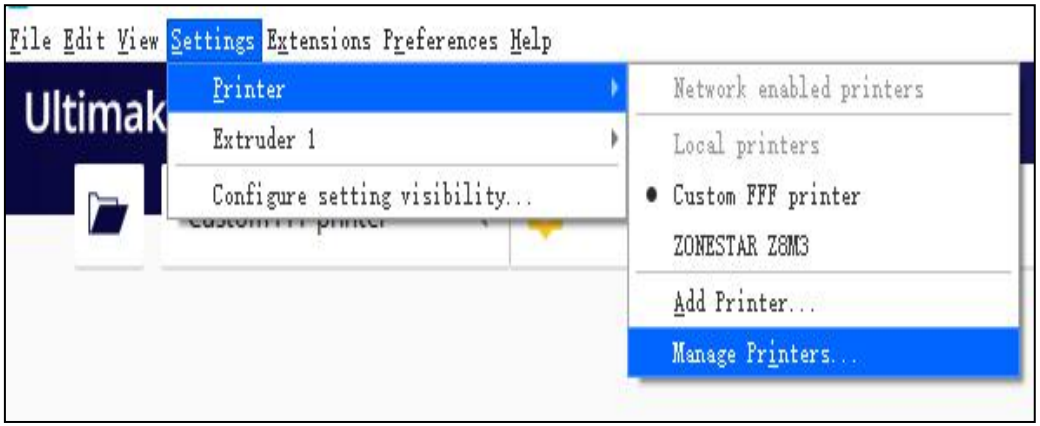




3. Run the cura

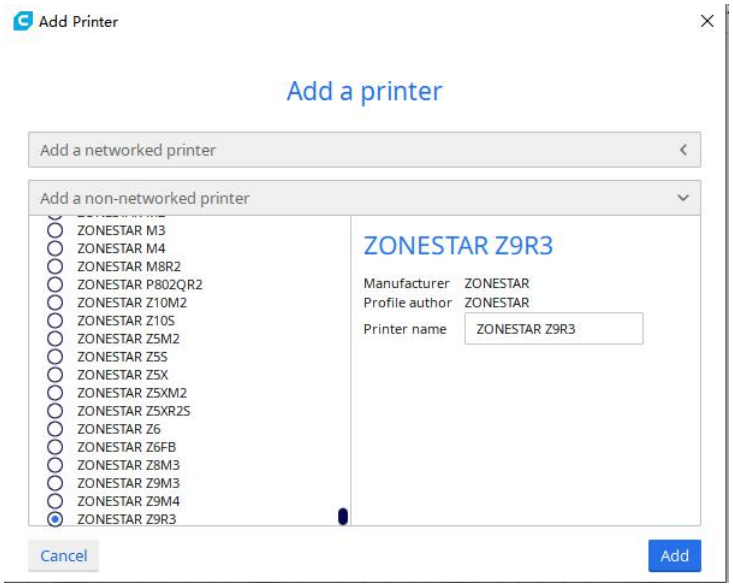
Setting up printer

1: Open “Manage printers”

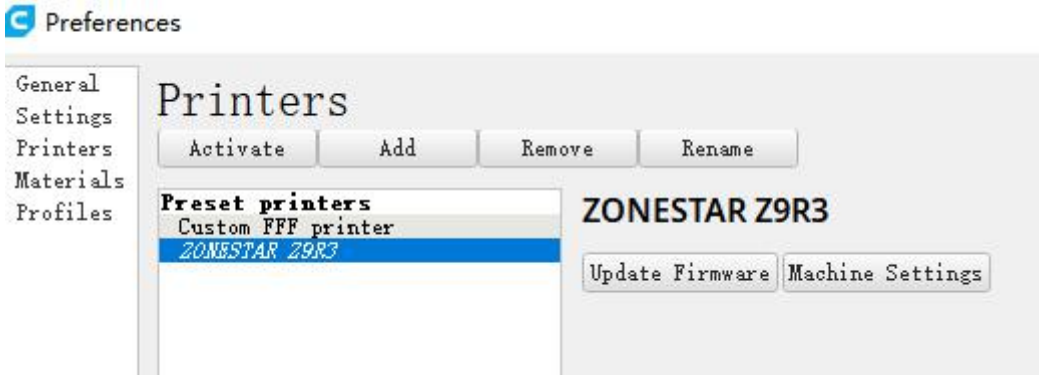


2: Click “Add”

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

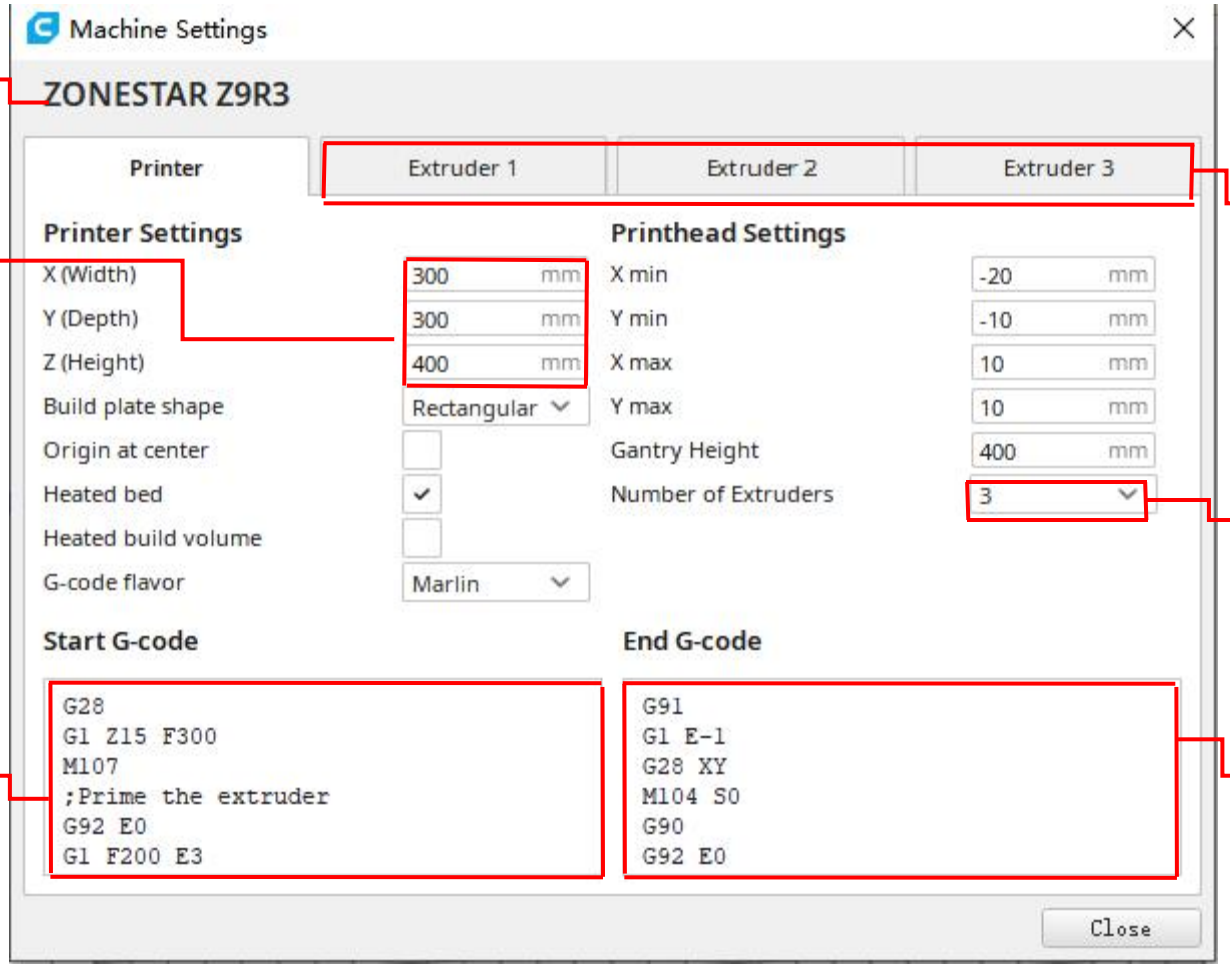


4: Click “Activate”



Setting up printer

Click “Machine settings”. Check the printer parameters.



Printer model

Print size

Start G-code

Extruder name

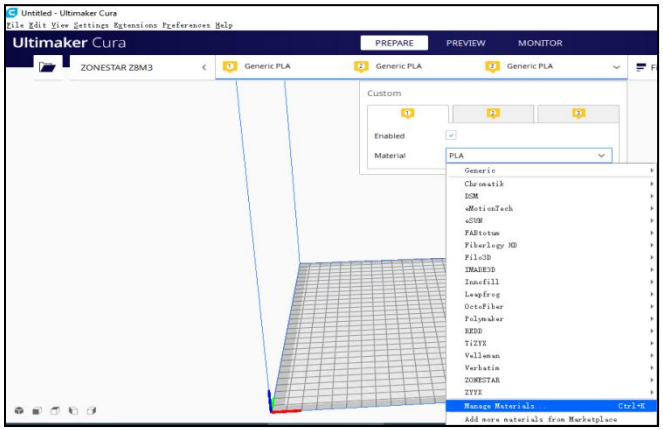
Number of extruders

End G-code

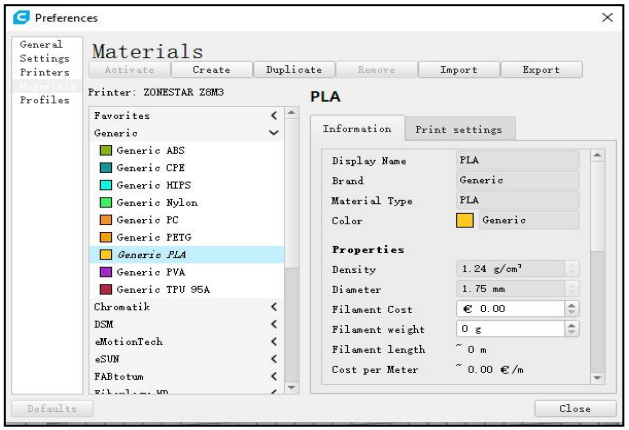
Setting up filament

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

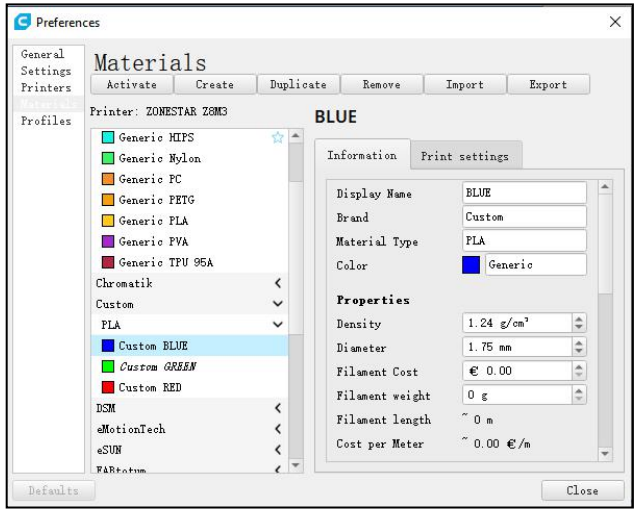
1: Open “Manage materials...”



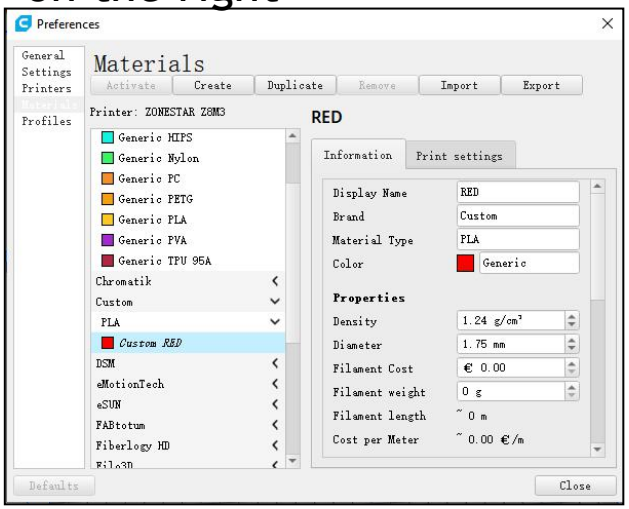
2: Click “Create”



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

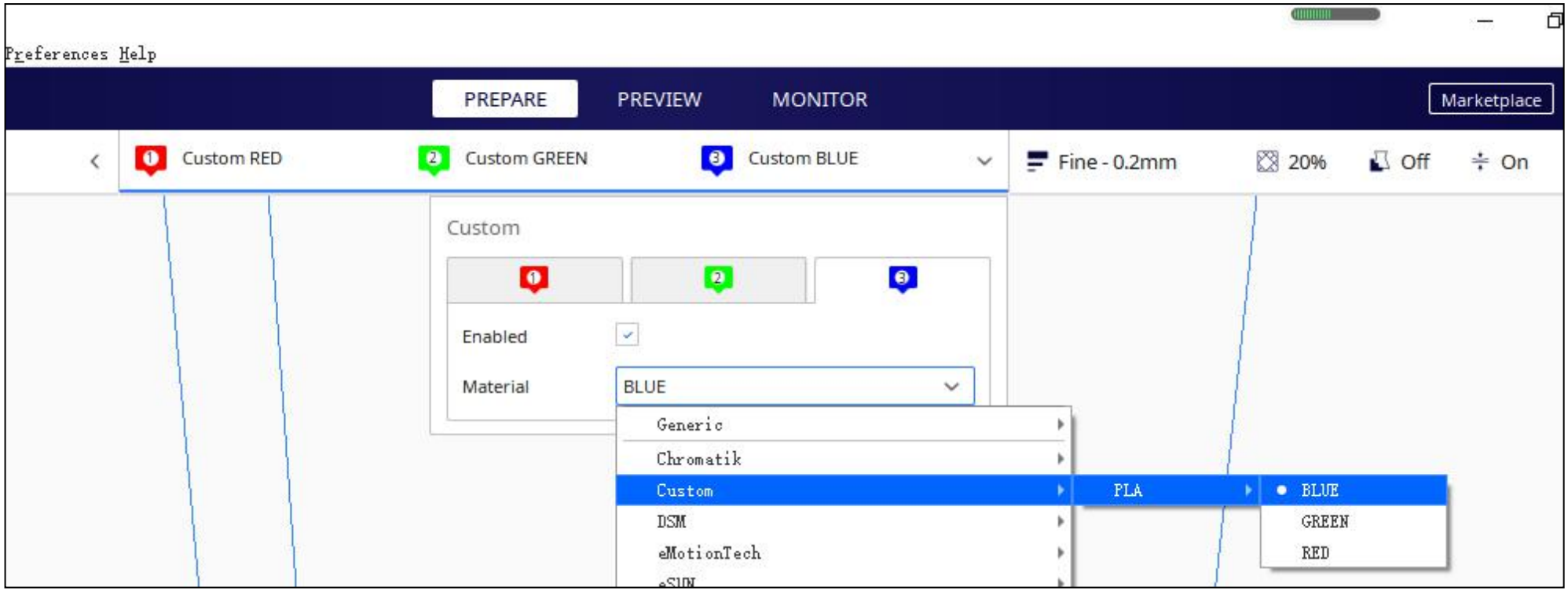


3:Find custom and modify relevant parameters in the information column on the right



Setting up filament

Select the filament color in turn on the main page.As shown in the figure below.



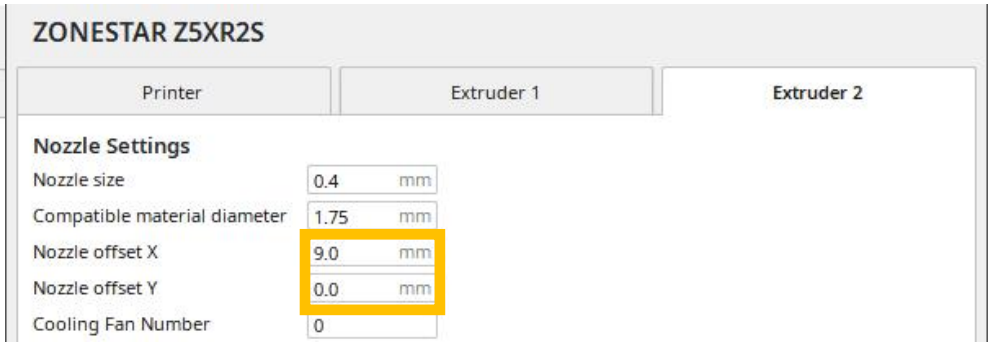
You can choose ZONESTAR filament too, we have set some color for PLA and PETG

Parameter setting of Extruder

For multicolor printing, the offset parameter setting of extruder is very important, otherwise, the printing model cannot be completed. For the R series multi-color printer of zonestar company, if you have integrated it in Cura software, you only need to select the corresponding printer model. Otherwise, you have to set the offset parameters of the extruder manually..

●Two color printing, note that the number of extruder must be consistent with the connection sequence on the motherboard.

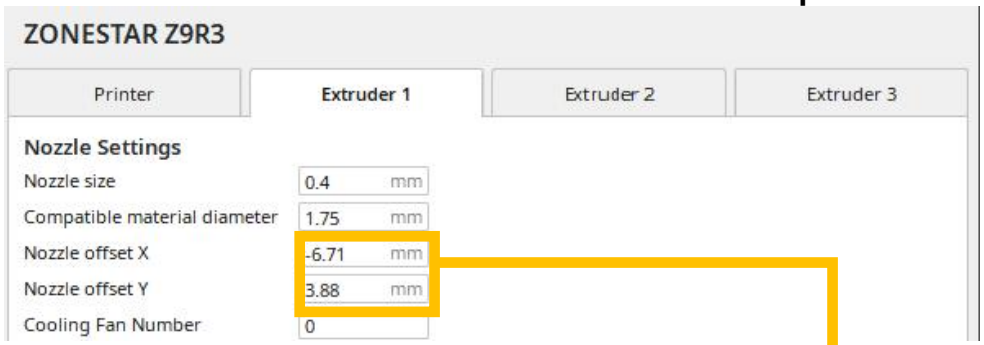
Extruder1

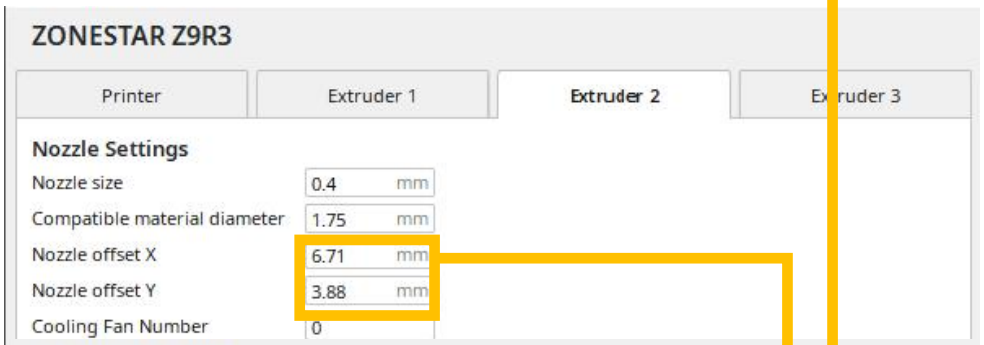


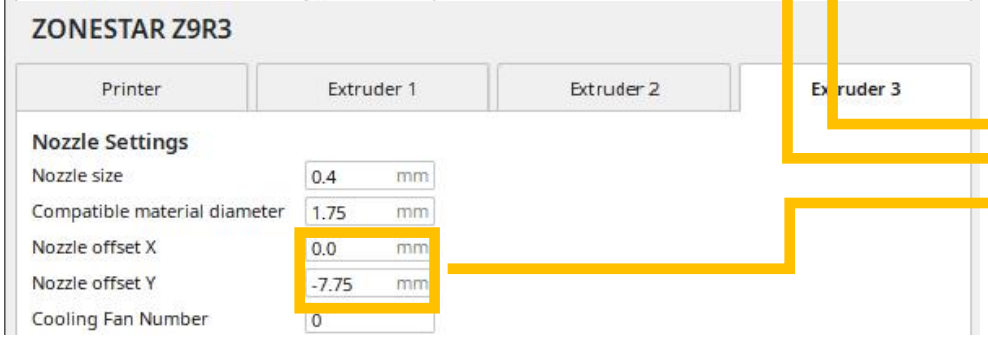
Extruder2

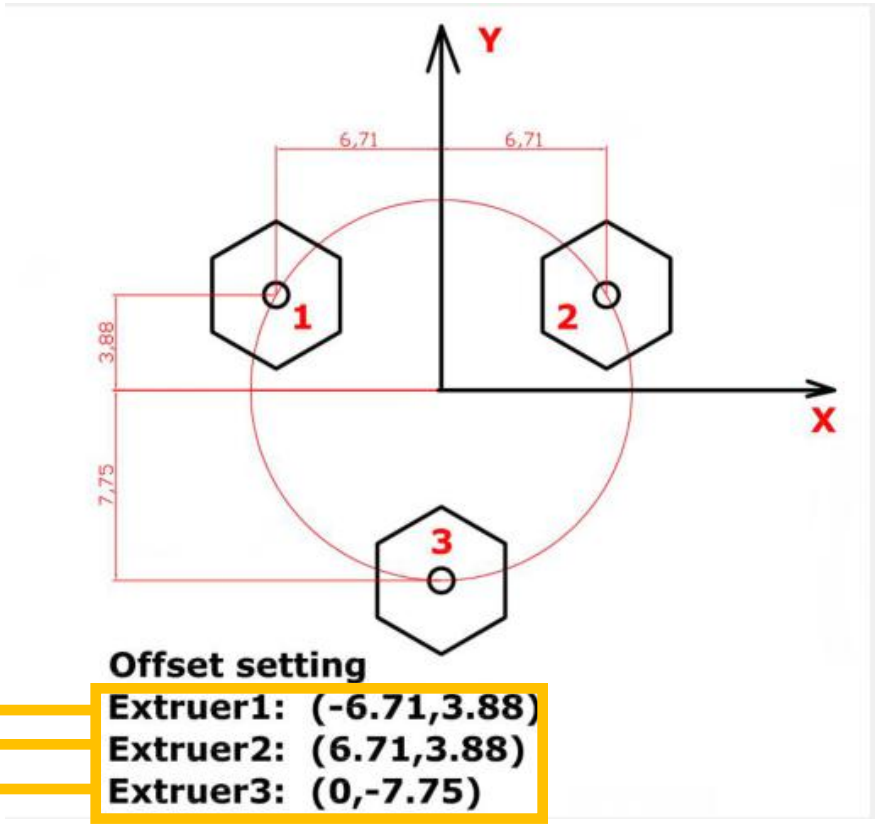
Parameter setting of Extruder

●Three color printing, as shown in the figure below. Note that the number of extruder must be consistent with the connection sequence on the main board.



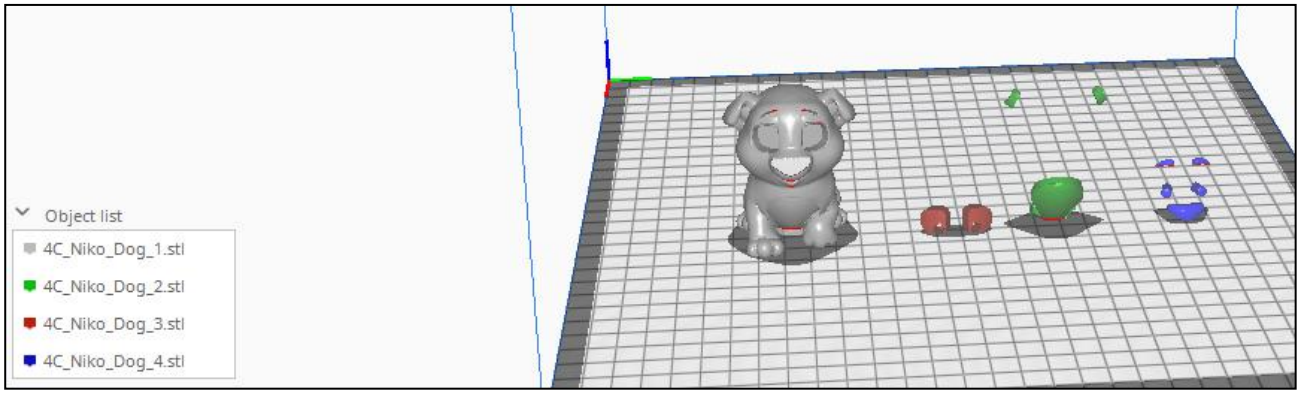






Slicing

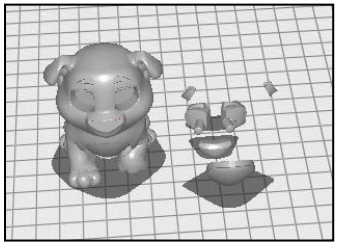
For the sake of illustration, we will use only one 3d object in the following pages. This 3d object is a 4-color model, which has divided the object into four parts

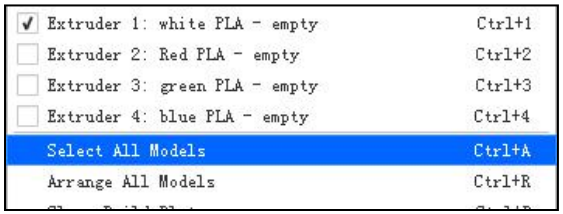


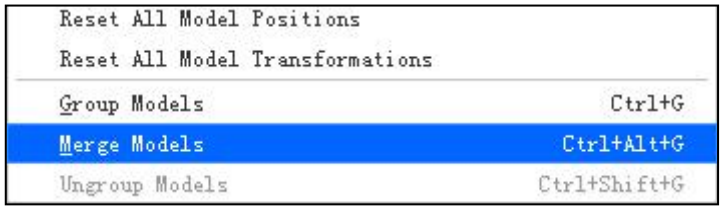
If you need to print multicolor, you need a 3d object that has been divided (the number of divided parts is according to the number of colors), and their origin position must be consistent in order to be merged.

Of course, you can also merge several objects into one color (multiple parts are assigned to the same extruder), as you will see in the next pages

Slicing singel color 3d object



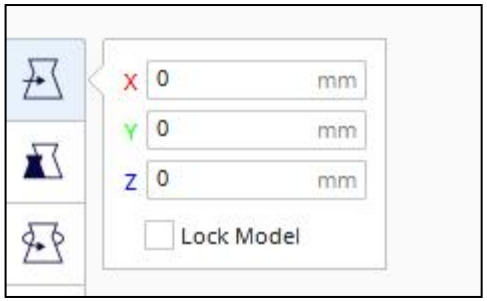


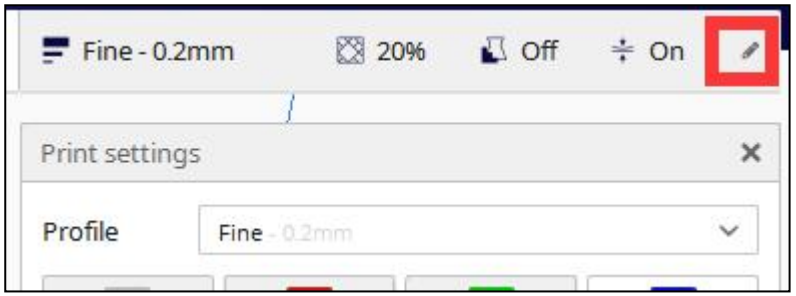


Load files

Right click the mouse
Setlect all models

Merge





Move/Scale/Rotat
e the model

Set slicing parameter

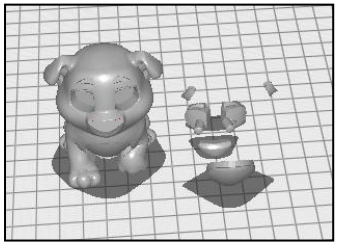
NOTE: load “1C_Niko_dog.3mf” to see reference settings

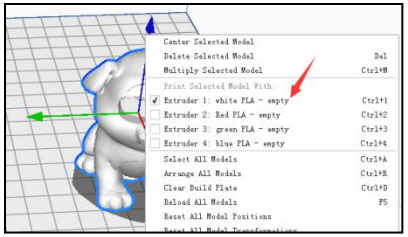
Slicing it

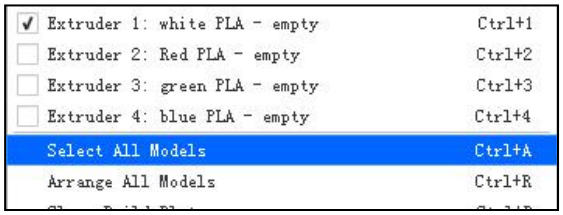
save it

Copy the gcode file to SD card, then print it

Slicing multi colors 3d object



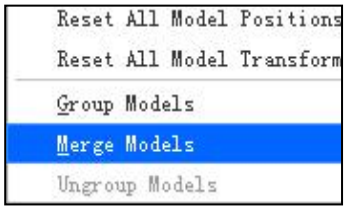


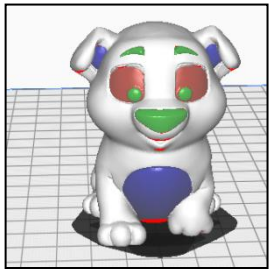


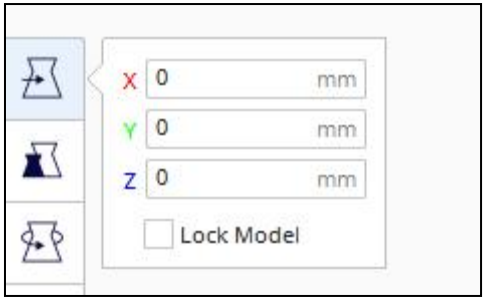
Load files

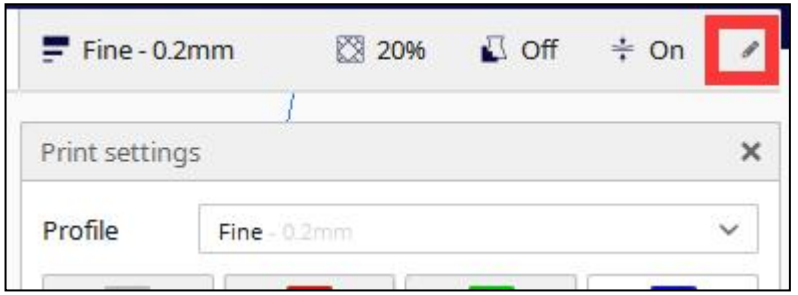
Right click the part and assign extruder for each

Right click the mouse Setlect all models





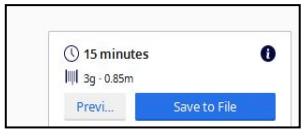


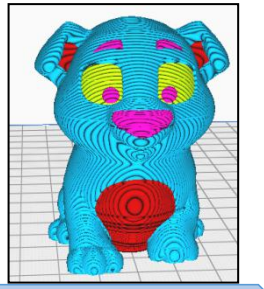


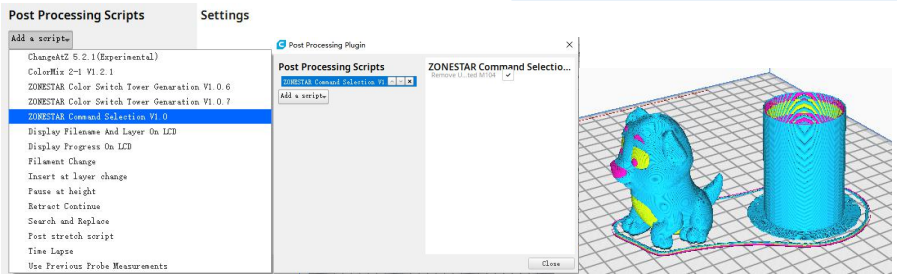
Merge

Move/Scale/Rotat e the model

Set slicing parameter







slicing and save it to PC

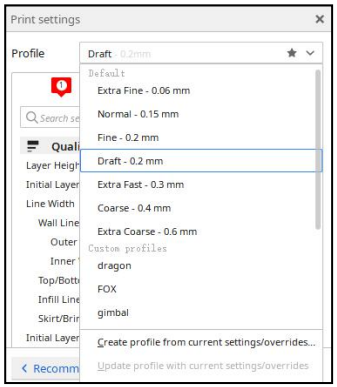
Add Command Selection Plugin

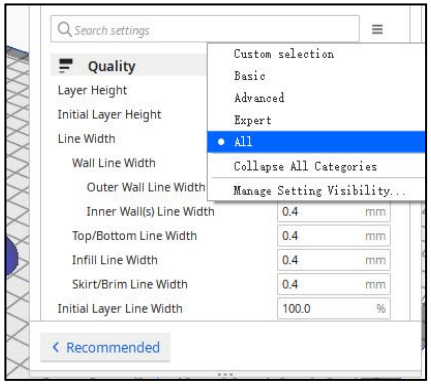
Copy to SD card and print it

Slicing multi colors 3d object - slicing

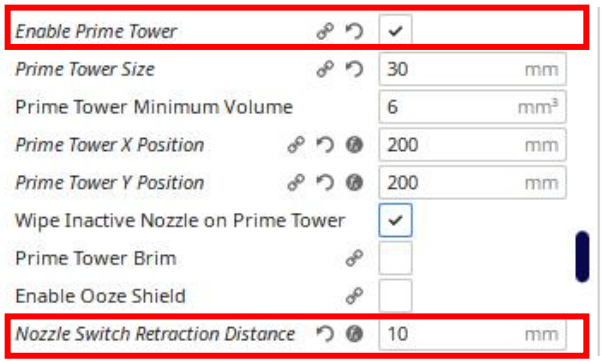
NOTE: When printing settings, please note that it needs to be set for each extruder.

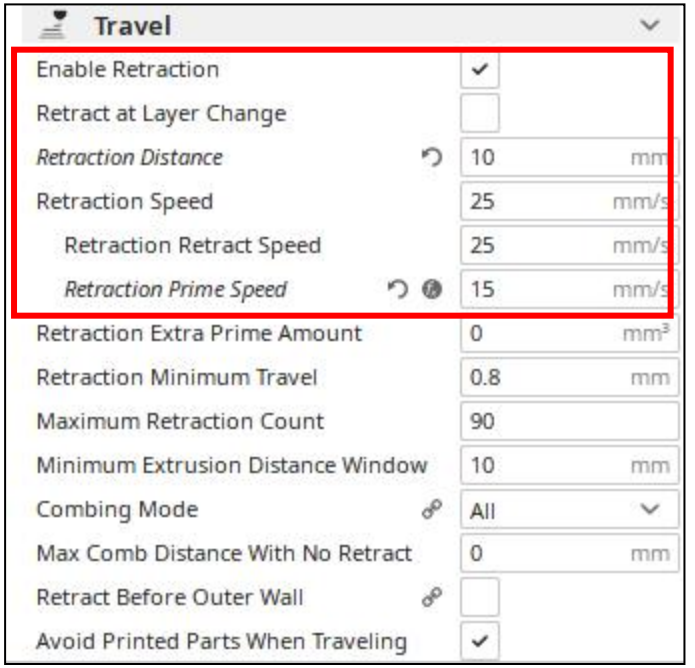
NOTE: The below settings are for PLA filament, if you want to choose other type of filament, please modify the nozzle temperature hotbed temperature to correct value

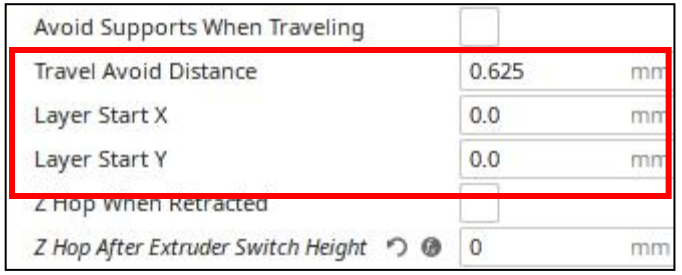




NOTE: Pay attention to the retraction parameters in "travel" and "dual Extrusion", as below:

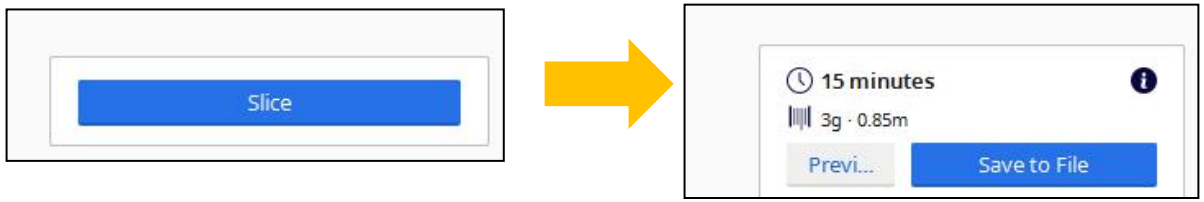




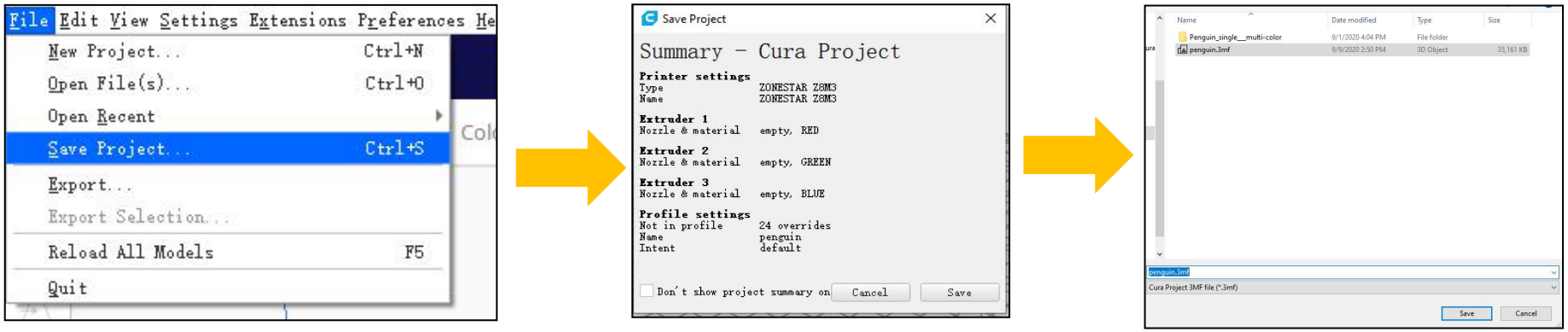


Slicing multi colors 3d object - Slice and Save

When all settings are completed, slicing and store the gcode file to your PC.

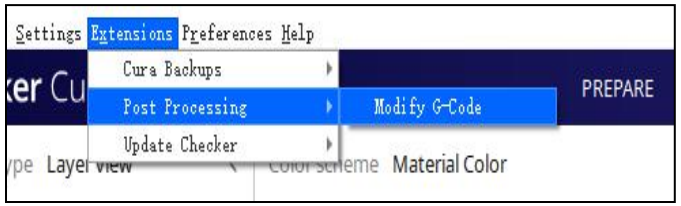


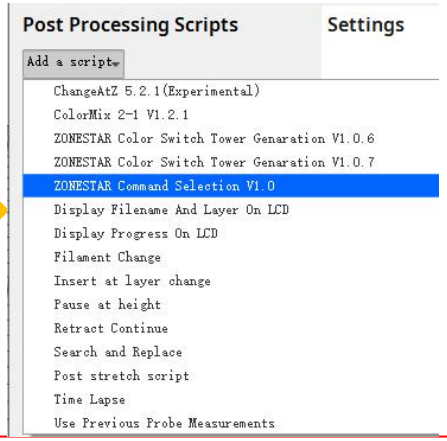
We suggest you save the project file for the next call. All configurations will follow the previous settings.

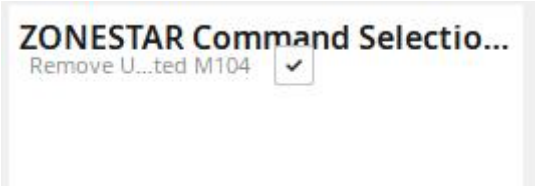


Slicing multi colors 3d object - Command Selection Plugin

1.Open plug-in.



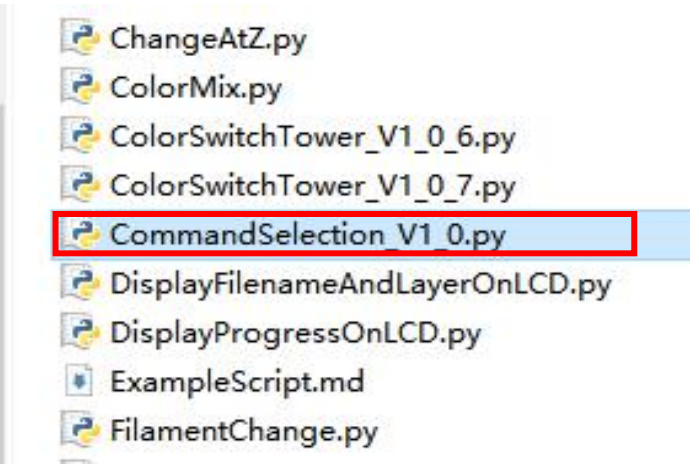




NOTE: If you can't find the "ZONESTAR Color Switch Tower Generation" plug-in in the list, please check if there is a "ColorSwitchTower.py" in the below directory of your PC:

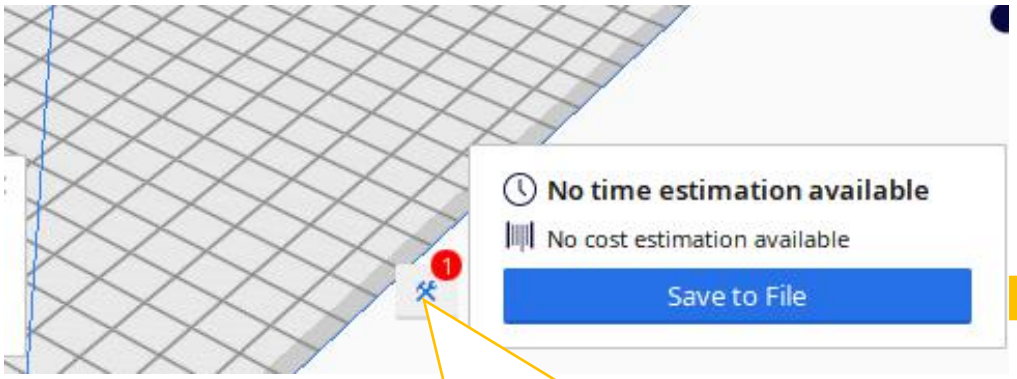
%install directory of Cura%\plugins\PostProcessingPlugin\scripts

If there isn't, please refer to "page 4" to download it and copy to the above directory.

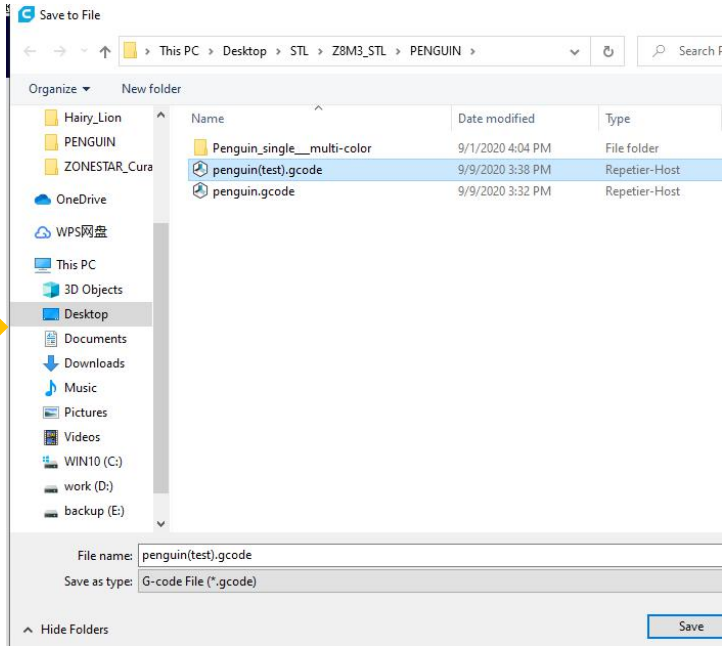


Slicing multi colors 3d object - Command Selection Plugin

2.Open the sliced gcode file(e.g.: dog.gcode) and then save it to a new gcode file (e.g.: dog(test).gcode).



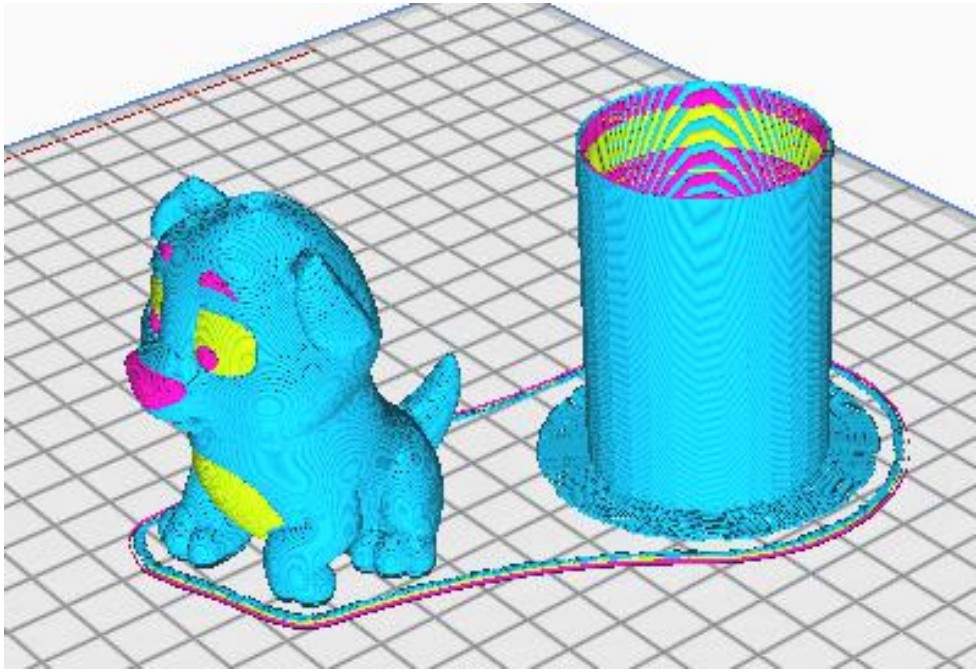
The plug-in tool is shown here.



When save the new gcode file, the plug-in will work automatically and add the color swithing tower to the new gcode file.

Slicing multi colors 3d object - Command Selection Plugin

Open the new gcode file (dog(test).gcode) you saved, preview it and check if it is on the right position.



NOTE:

Due to Cura software, the temperature will be reset when switching the extruder in gcode file after polychromatic slicing of the model, which may cause the printing to be unable to be completed. Therefore, the gcode file must be re opened, and the plug-in must be added to save it again.