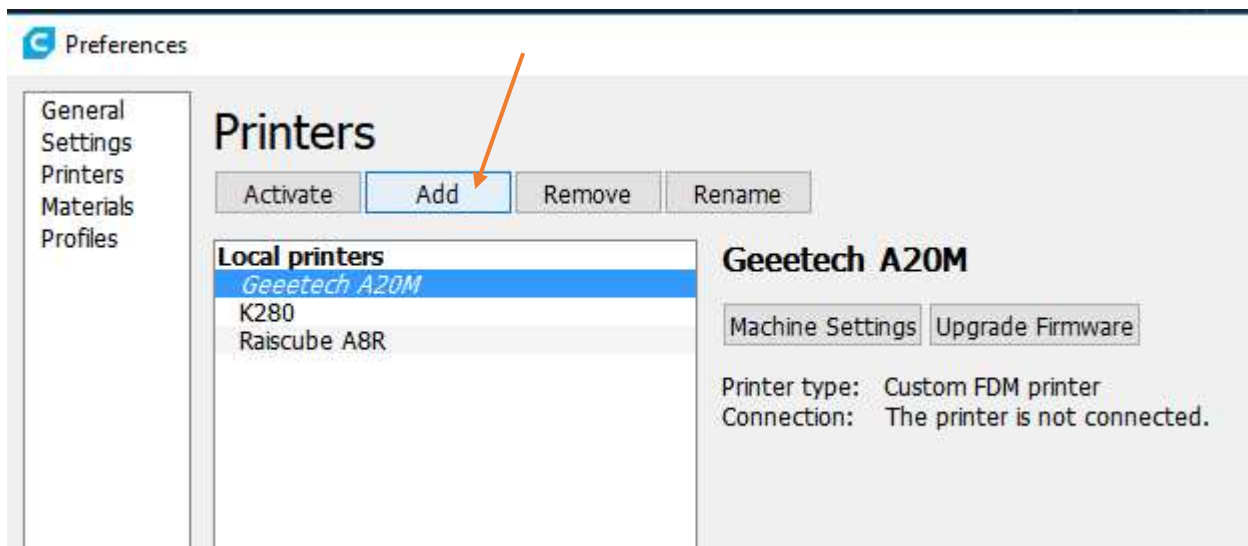
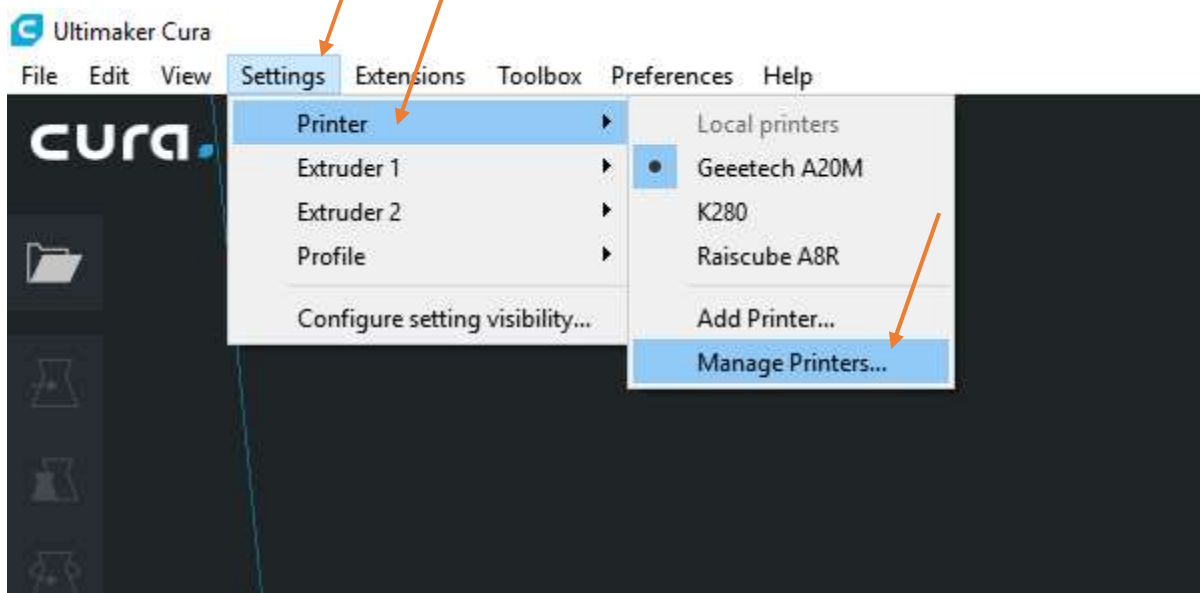



Guide for Cura 3.6.0 and the quirks with using it.

First thing to do is setup printer



 Add Printer

> **Ultimaker**


> **Custom**

☒ Custom FDM printer

> **Other**

Printer Name:

Add Printer



Setting up 3 virtual extruders. Extruders 3-5 are virtual with mixes 75/25 50/50 and 25/75 respectively.

Machine Settings

Printer Extruder 1 Extruder 2 Extruder 3 Extruder 4 Extruder 5

Printer Settings

X (Width) 255 mm

Y (Depth) 255 mm

Z (Height) 255 mm

Build plate shape Rectangular

☐ Origin at center

☒ Heated bed

G-code flavor Marlin

Start G-code

```
M163 S0 P0.75
M163 S1 P0.25
M164 S2
M163 S0 P0.50
M163 S1 P0.50
M164 S3
M163 S0 P0.25
M163 S1 P0.75
M164 S4
T0
M109 S{material_print_temperature};
M104 T{initial_extruder_nr} S{material_print_temperature};
G28 ;Home
G1 Z15.0 F6000 ;Move the platform down 15mm
G92 E0
G1 F200 E3
G92 E0
```

Printhead Settings

X min 20 mm

Y min 10 mm

X max 10 mm

Y max 10 mm

Gantry height 9999999999mm

Number of Extruders 5

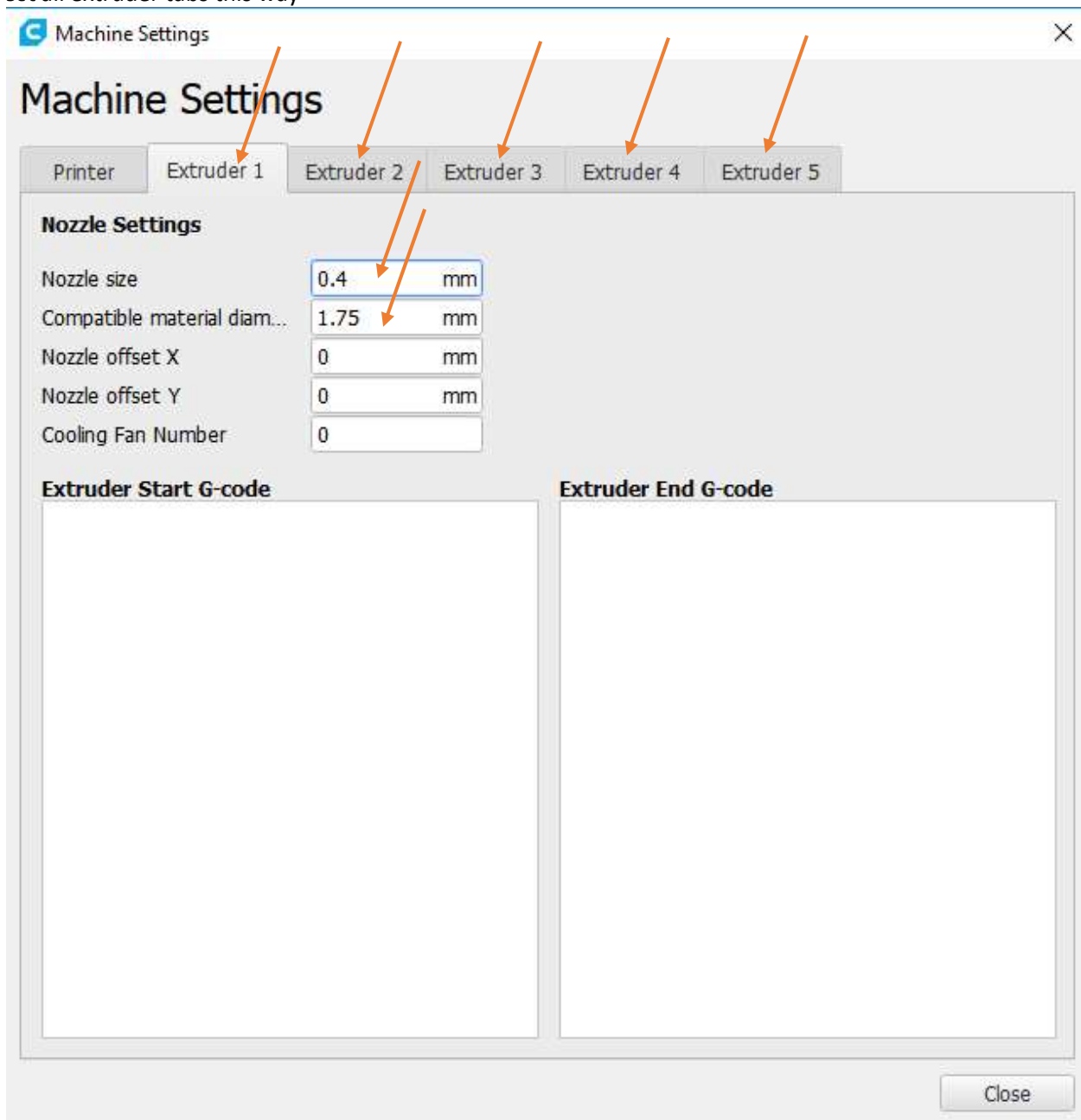
End G-code

```
M104 S0
M140 S0
;Retract the filament
G92 E1
G1 E-1 F300
G28 X0 Y0
M84
```

Close

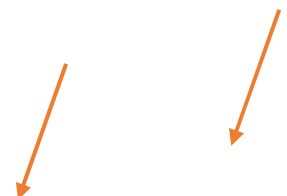
```
M163 S0 P0.75
M163 S1 P0.25
M164 S2
M163 S0 P0.50
M163 S1 P0.50
M164 S3
M163 S0 P0.25
M163 S1 P0.75
M164 S4
T0
M109 S{material_print_temperature};
M104 T{initial_extruder_nr} S{material_print_temperature};
G28 ;Home
G1 Z15.0 F6000 ;Move the platform down 15mm
G92 E0
G1 F200 E3
G92 E0
```

set all extruder tabs this way



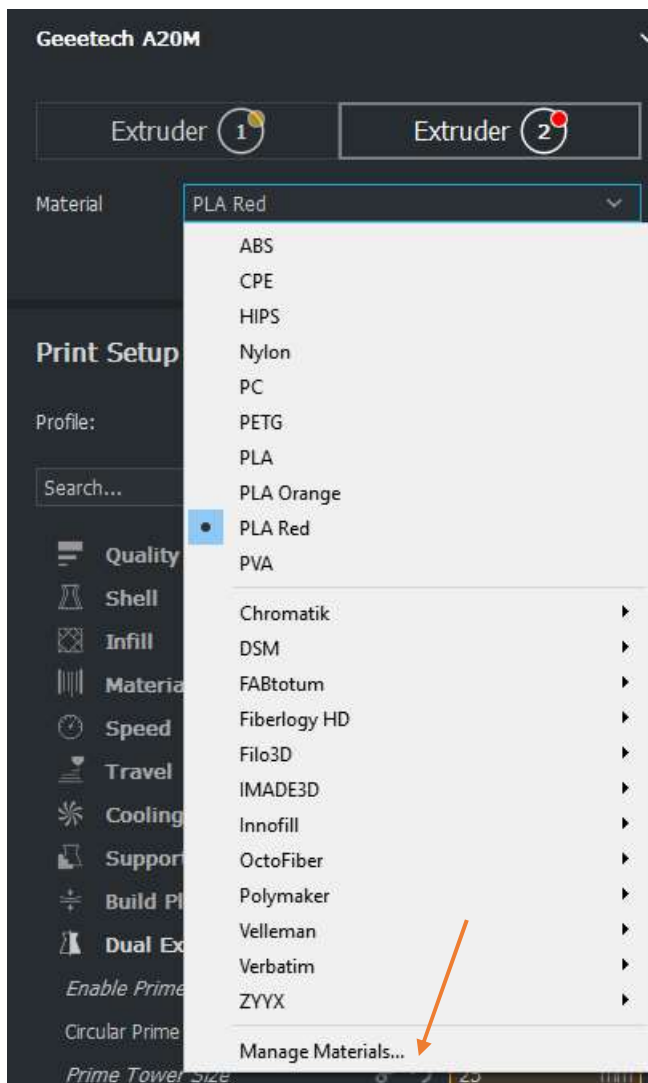
Close then close

Set your printer to A20M

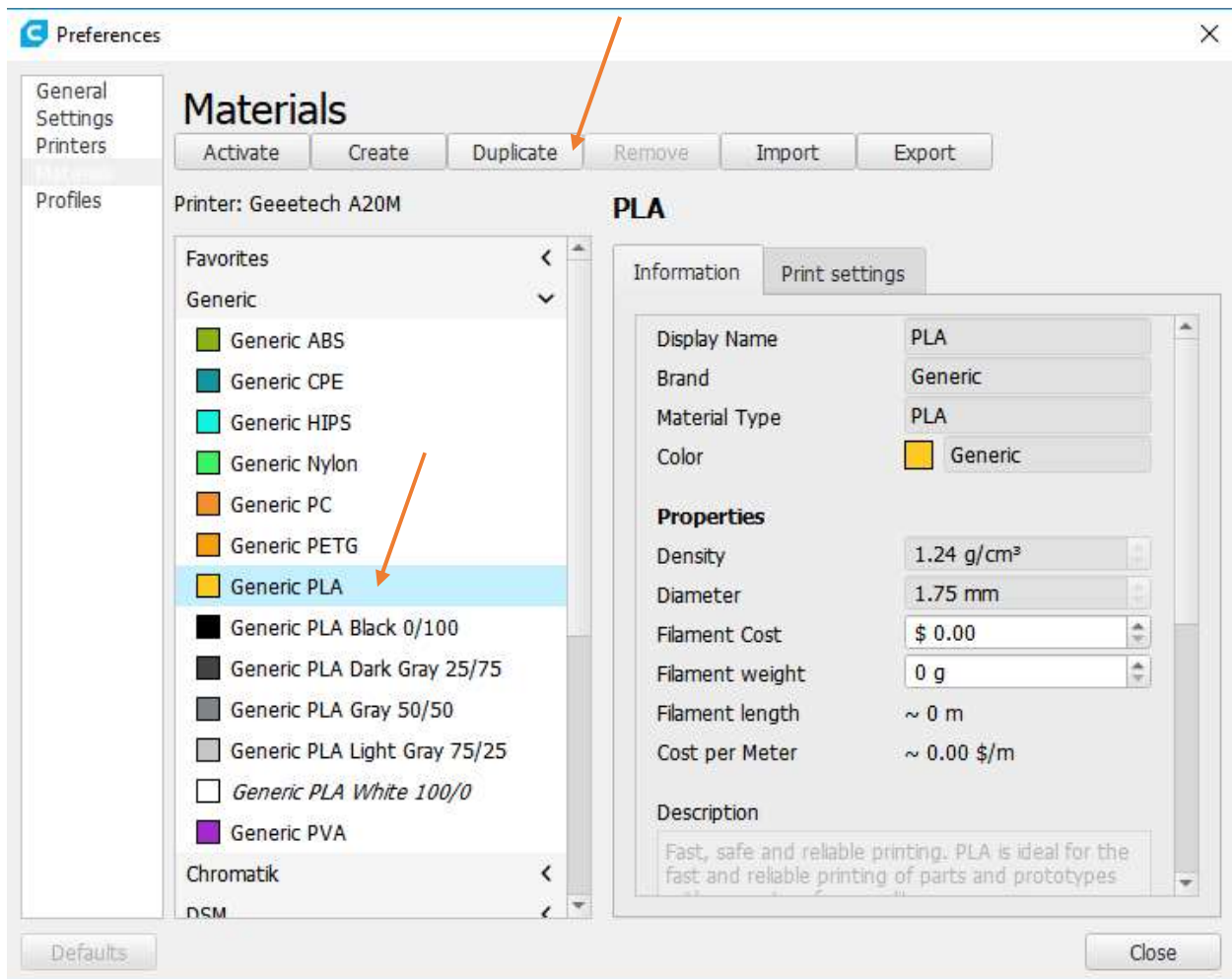


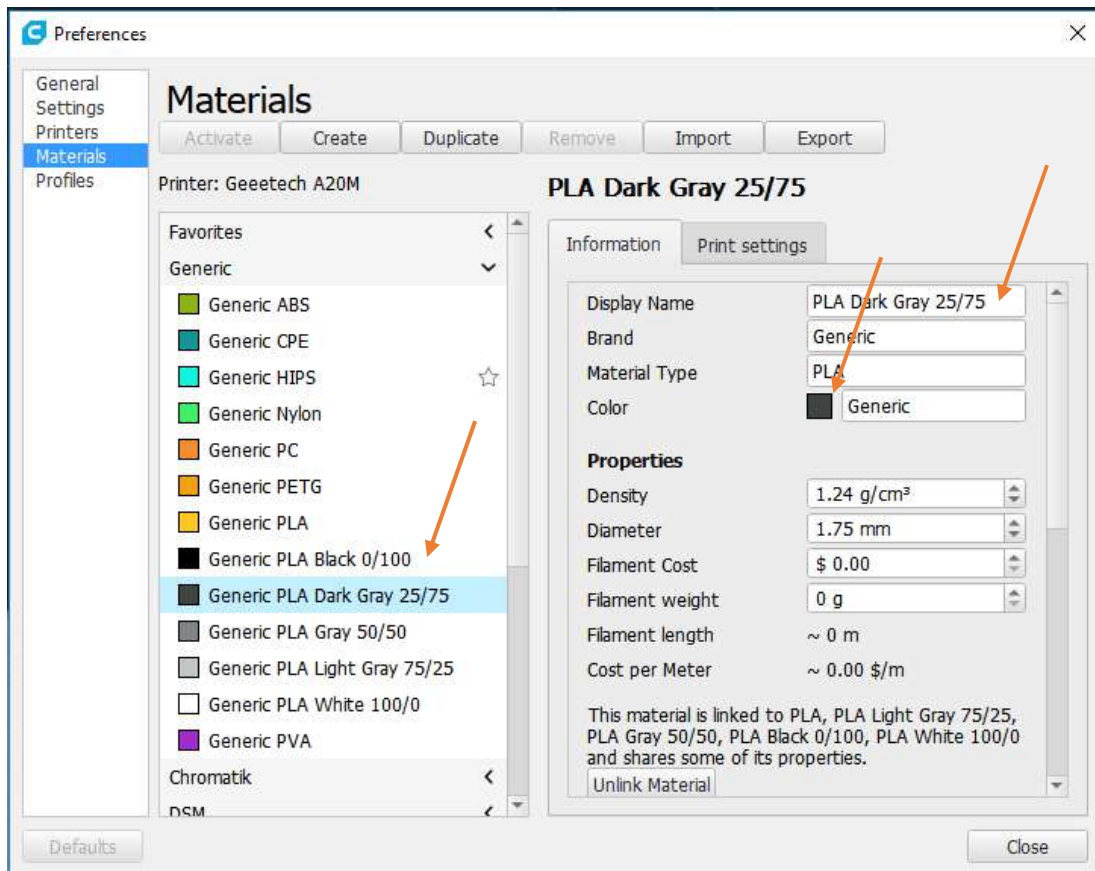


Manage Materials



Highlight Generic PLA and duplicate 5 times





Change the display name and the color. In this example I used white, black, light gray, gray , and dark gray. The numbers represent the mixture.

1st number is extruder 1 percentage and the 2nd is extruder 2. These should match the start gcode above.

Now assign the extruders the appropriate colors

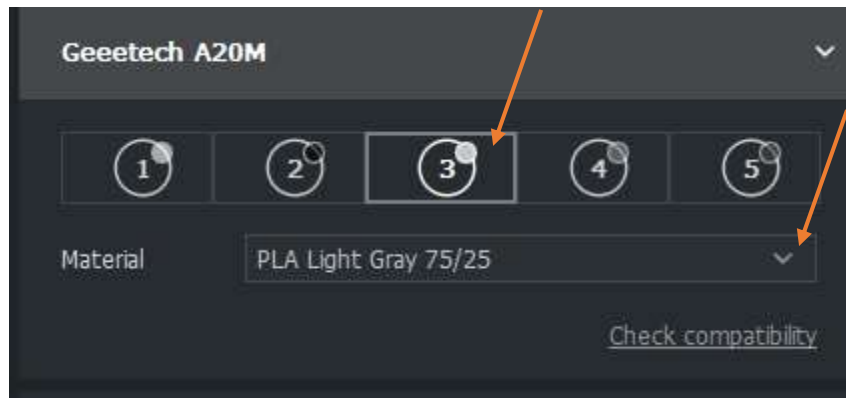
1 White

2 Black

3 Light Gray

4 Gray

5 Dark Gray

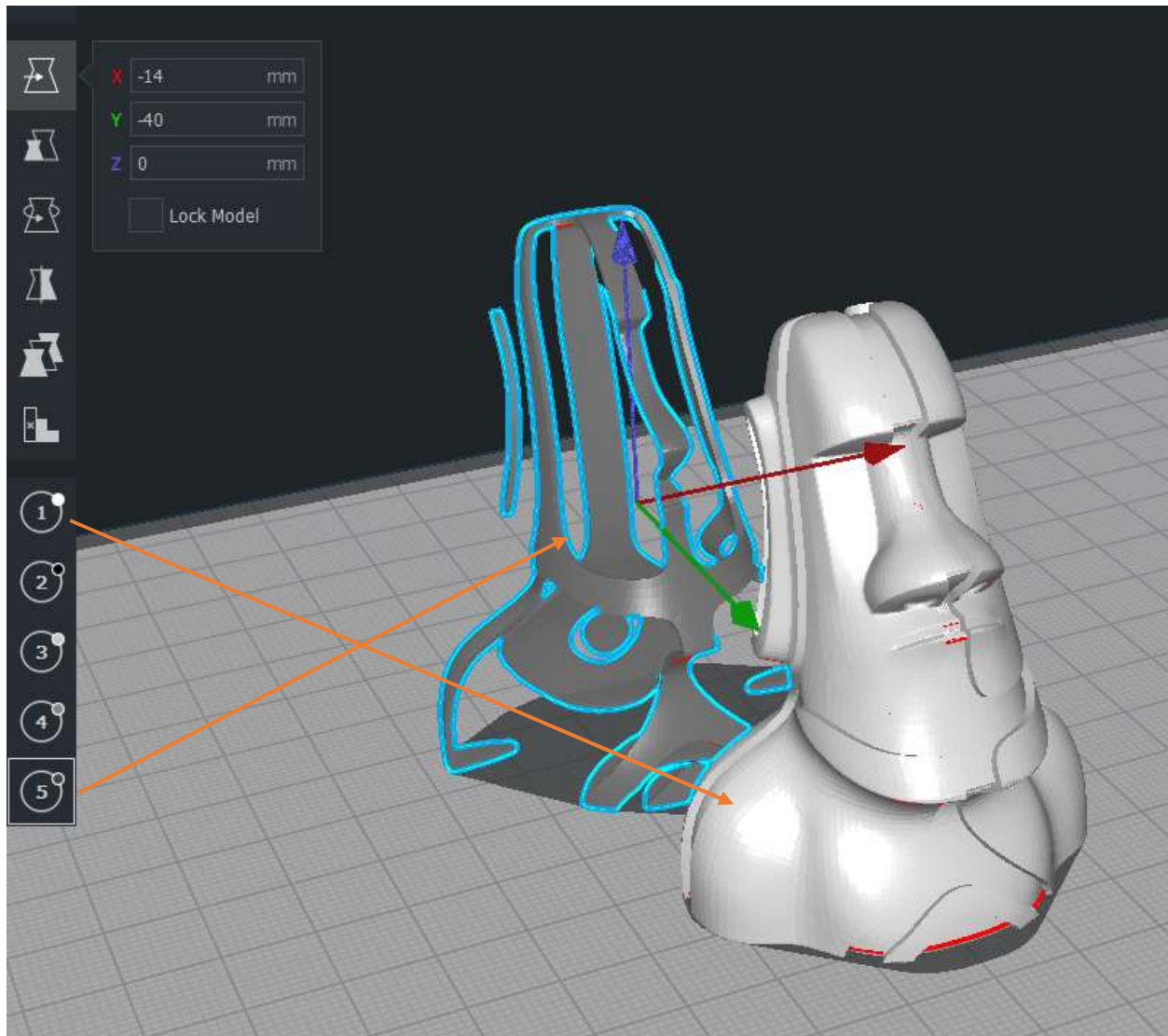


Drag and drop your print into Cura separate models show up.

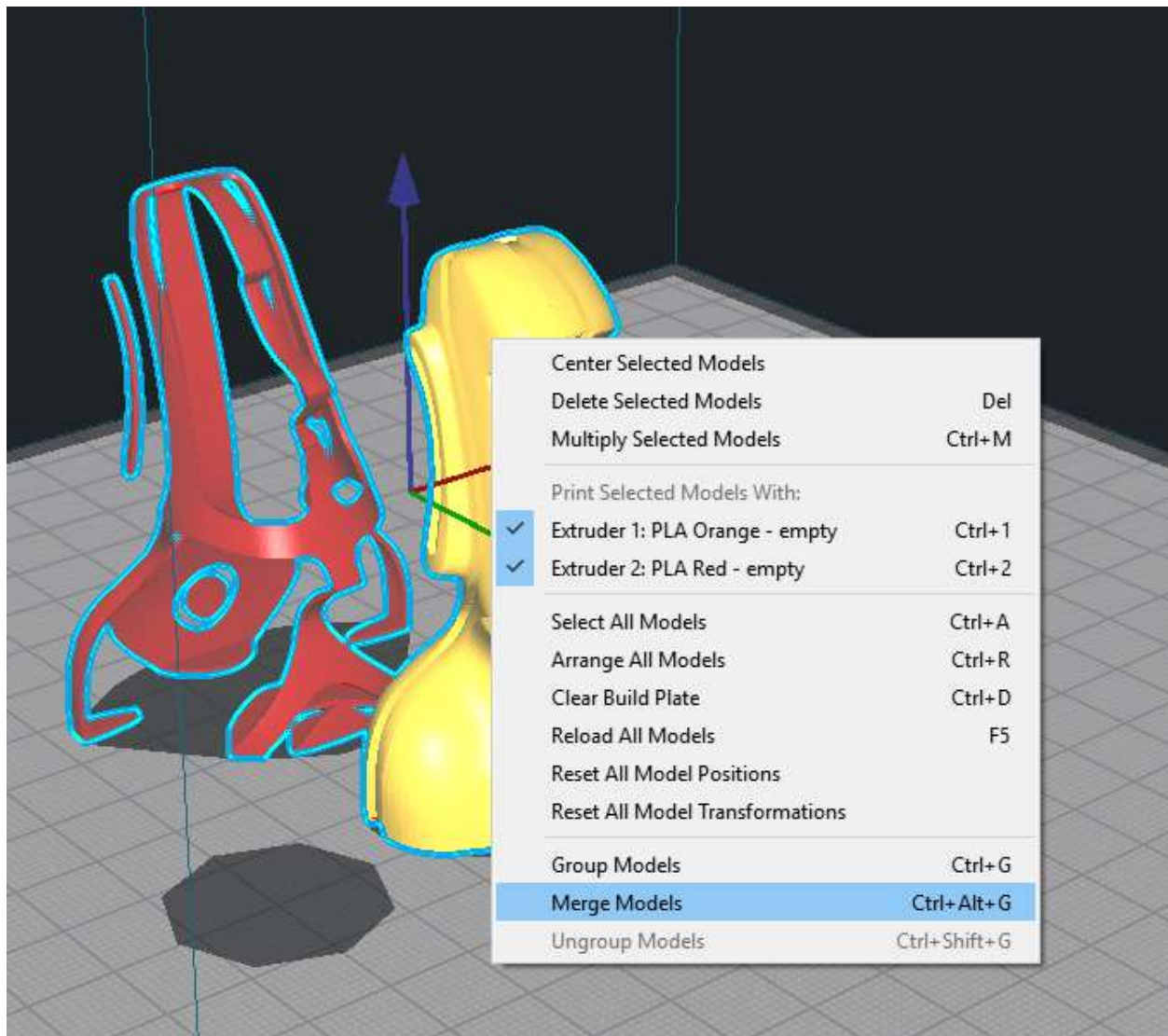
Assign Each model an extruder

Click the model then the extruder. Then click the other model and assign the other extruder

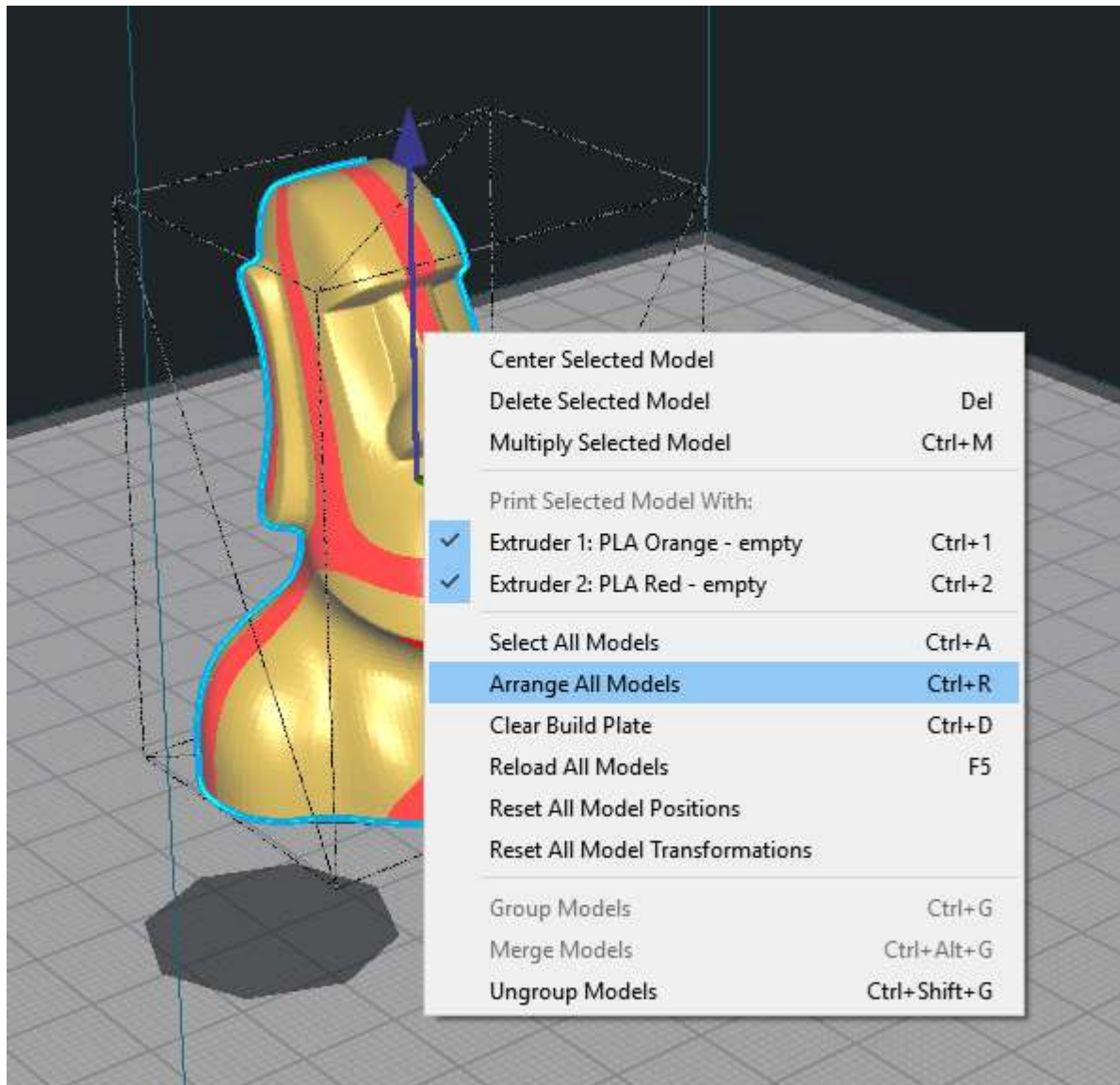
The colors should correspond



Grab both models by left clicking 1 of them then do a cntrl A or left click 1 then shift left click the other
Right click then Merge models



Right click on the model then Arrange all models to center the model



For each extruder at the top you will need to set all these settings.... If settings are not shown it is because they are default. You can do a copy by right clicking on each one changed and copy to all extruders but please check. This is a pain in the butt and I hope Cura can support a better copy. It only works on the Sub category you right click on.

Look for the  to indicate it was changed



Material

Default Printing Temperature

215

°C

Printing Temperature

215

°C

Printing Temperature Initial Layer

215

°C

Initial Printing Temperature

215

°C

Final Printing Temperature

215

°C

Default Build Plate Temperature

70

°C

Build Plate Temperature

70

°C

Build Plate Temperature Initial Layer

70

°C

Flow

110

%

Initial Layer Flow

110

%

Enable Retraction

☒

Retract at Layer Change

☐

Retraction Distance

6

mm

Retraction Speed

60

mm/s

Retraction Retract Speed

60

mm/s

Retraction Prime Speed

60

mm/s

Retraction Extra Prime Amount

0

mm³

Retraction Minimum Travel

0.8

mm

Maximum Retraction Count

90

Minimum Extrusion Distance Window

6

mm

Standby Temperature

215

°C

Nozzle Switch Retraction Distance

16

mm

Nozzle Switch Retraction Speed

60

mm/s

Nozzle Switch Retract Speed

60

mm/s

Nozzle Switch Prime Speed

60

mm/s

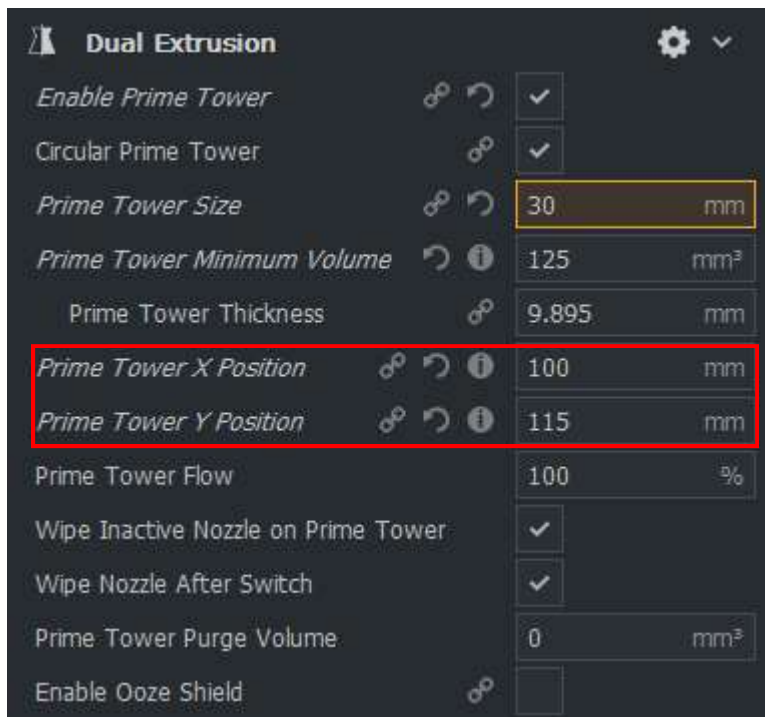
Speed
<

<i>Print Speed</i>		45	mm/s
Infill Speed		45	mm/s
Wall Speed		22.5	mm/s
Outer Wall Speed		22.5	mm/s
Inner Wall Speed		45.0	mm/s
Top/Bottom Speed		22.5	mm/s
<i>Travel Speed</i>		150	mm/s
Initial Layer Speed		22.5	mm/s
Initial Layer Print Speed		22.5	mm/s
Initial Layer Travel Speed		75.0	mm/s
Skirt/Brim Speed		22.5	mm/s
Maximum Z Speed		0	mm/s
Number of Slower Layers		2	
Equalize Filament Flow		<input type="checkbox"/>	
Enable Acceleration Control		<input type="checkbox"/>	
Top Surface Skin Acceleration		3000	mm/s ²
Enable Jerk Control		<input type="checkbox"/>	

Cooling
▼

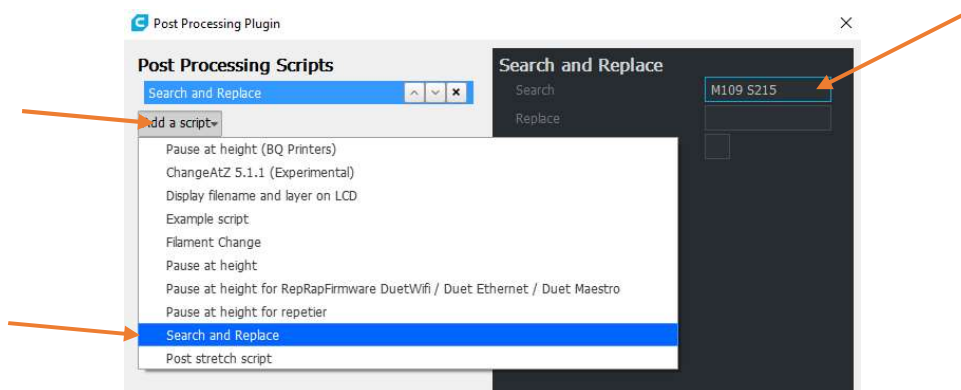
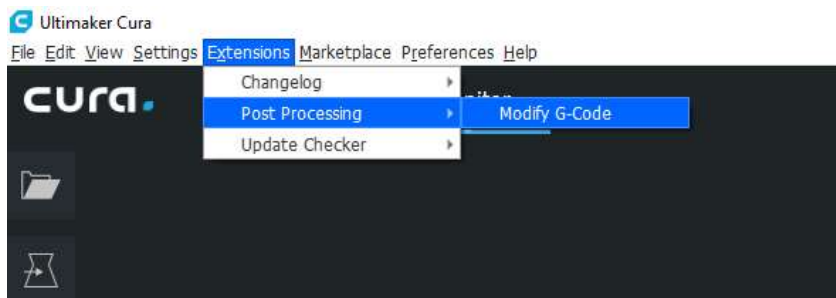
Enable Print Cooling		<input checked="" type="checkbox"/>	
<i>Fan Speed</i>		50	%
Regular Fan Speed		50	%
Maximum Fan Speed		50	%
Regular/Maximum Fan Speed Threshold		10	s
Initial Fan Speed		0	%
Regular Fan Speed at Height		0.3	mm
Regular Fan Speed at Layer		2	
Minimum Layer Time		5	s
Minimum Speed		10	mm/s
Lift Head		<input type="checkbox"/>	

You will need to move your prime tower to an appropriate place on each print. You can see the shadow



Oh course you need to adjust some of these settings to suit your needs.

The prime tower is large and I did this to purge the cooking filament in the hotend.



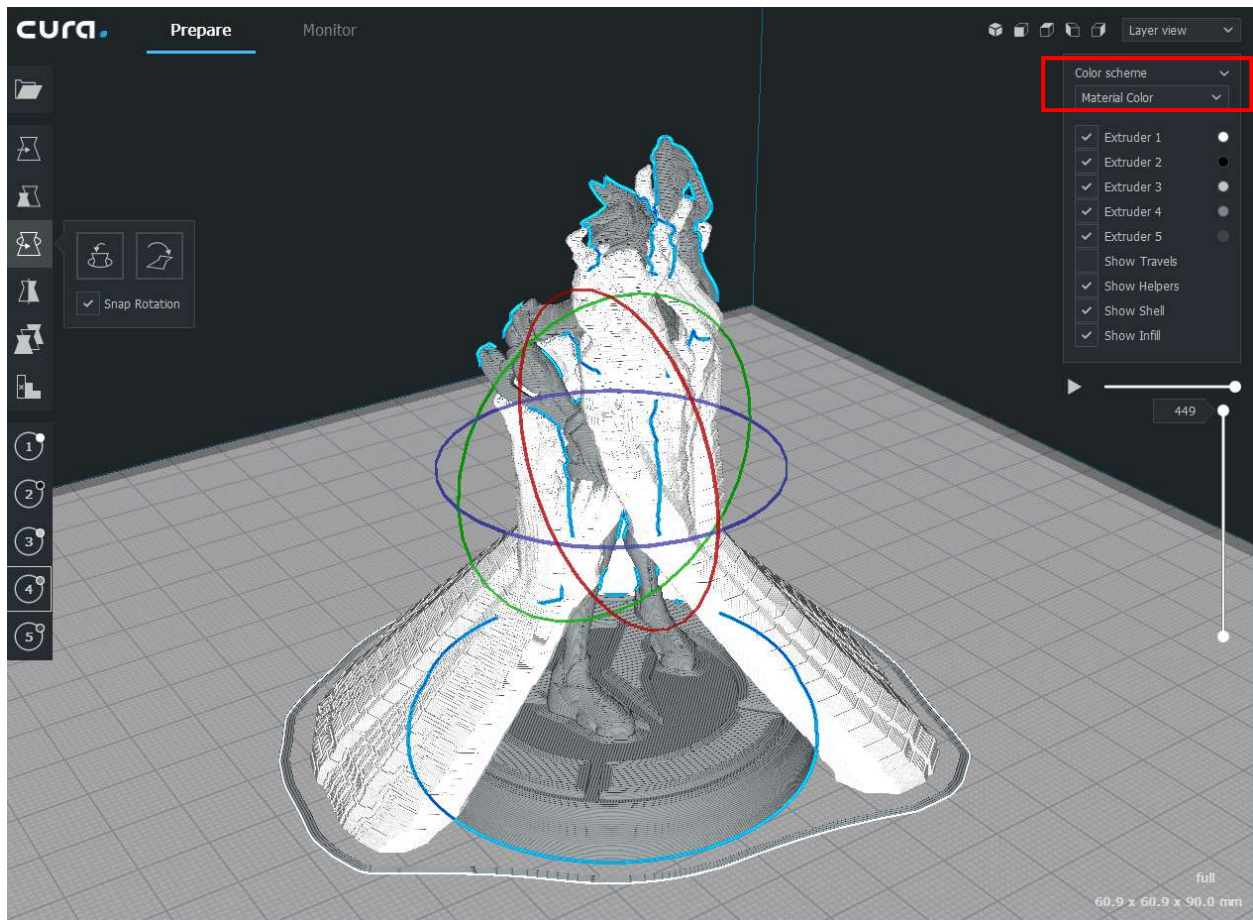
Remember we put 2 spaces in the start gcode above. Only put 1 space here.

Warning your display may report the wrong first percentage while warming up until it starts printing

Important

If you set a model to an extruder other than 1 to make sure build plate adhesion/supports/anything you can assign to a different extruder gets changed to that extruder as well unless you are using a purge tower.

You will need to change the purge tower size accordingly.



Here is a quick way to see if your extruders are all set. Set the color scheme to material color and you can see the model is set to a different color than the support. Sometimes this is desirable, but you really need a prime tower in that situation.