

Printing multiple colors on the same layer using PrusaSlicer and Prusa Printers

This small write-up will try to explain how to print multiple colors on the same layer height using PrusaSlicer. The PrusaSlicer Version used in this tutorial is V2.0 on Windows.

- **Why the hell would I want to do that?**

Well, if you want a Text on the bottom layer to be in a different color instead of just embossed (better readability) or just add a little color to your designs

- **It is probably overly complicated?**

Not really..... I managed to do it, so why wouldn't you 😊

- **Why buy an MMU if I can do this as well?**

I don't know about you, but whilst it is perfectly ok to do a few manual color changes, I wouldn't see myself doing 500+ over 24 hours for a complex print 😊 😊 😊

- **What do I need to get started?**

Basically just a model designed by you or downloaded, PrusaSlicer, a little bit of configuration and some patience for the color changes

1) Create or download your model.

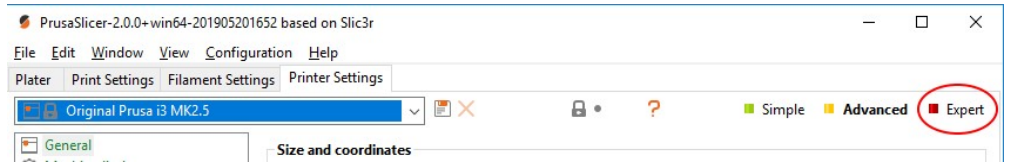
Basically, you need as many parts as you want colors. In case of the multi-colored Text for instance, you need the part itself, with the letters as "holes" and you need a second part, consisting of the letters at the same size and correct height. Here, as an example, you can see the Smiley I designed for this write-up (2 independent STL's) loaded into PrusaSlicer



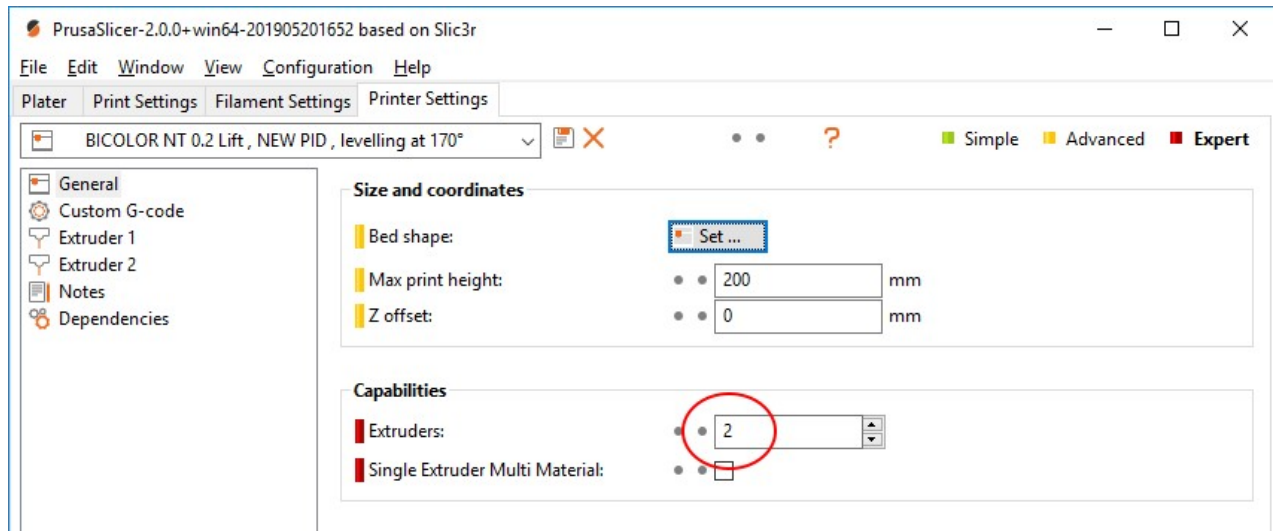
Correct Alignment of the Stl's depends on whether the parts were designed independently or together, they may open perfectly aligned or not, I will cover that later in this write-up !

2) Create a multi-color printer profile in PrusaSlicer

To do that, open up PrusaSlicer, switch to Expert mode (Multiple Extruder Settings are unavailable in Simple or Advanced Mode)

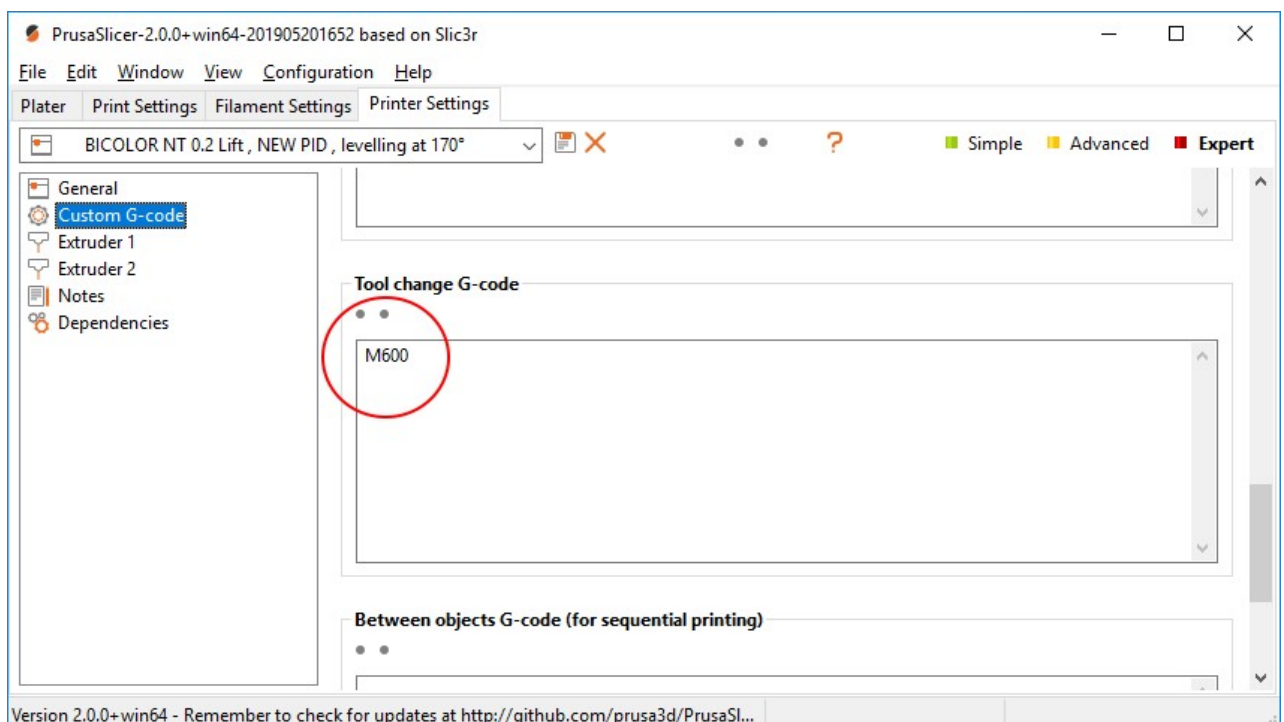


Then load your usual Printer Settings, and specify the number of extruders (colors) that you want to use in the general settings.



There is no need to check the "Single Extruder Multi material checkbox as we will do the color changes and purges manually anyway.

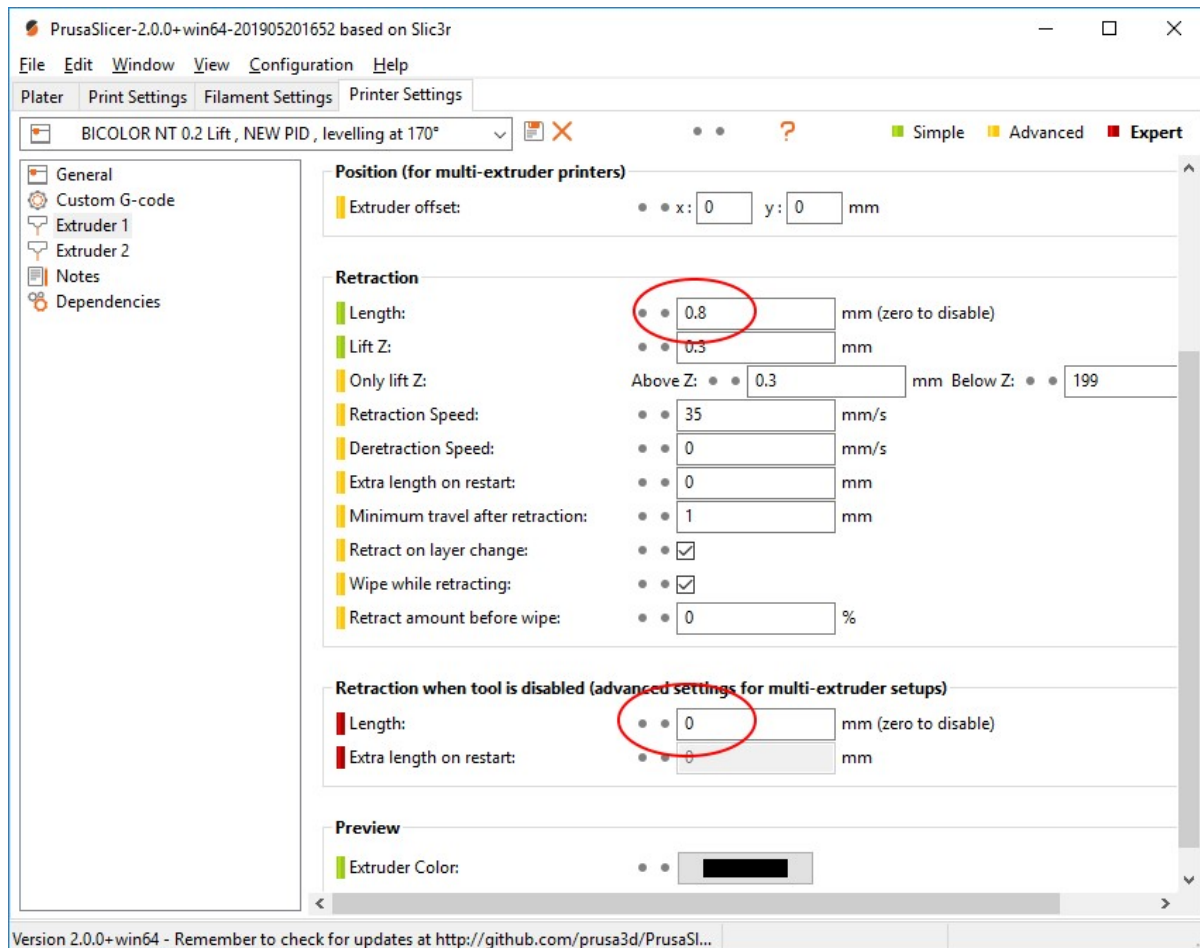
Then, in the Custom G-code Window, scroll down and set the Tool change G-code to be **M600**



Please also check **AND ADJUST** the retraction settings for **ALL** the extruders you configure.

Especially the “**Retraction when tool is disabled**” settings defaults to 4mm, which **WILL** cause blobs at every color change and is very likely to cause clogs! You **HAVE** to set that value set to **0** or you will run into problems.

Also the retraction length should not be too high ... I have found a value of 0.8 for general Retraction length to work well.



Again, you need to do that for **EVERY** extruder configured.

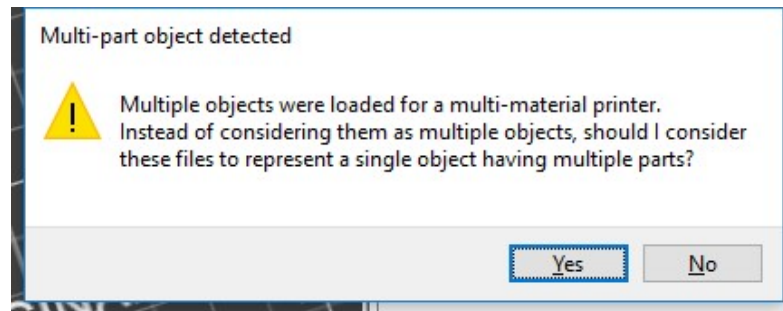
So much for the basic configuration, save those settings and go back to the plater

3) Load your model into PrusaSlicer

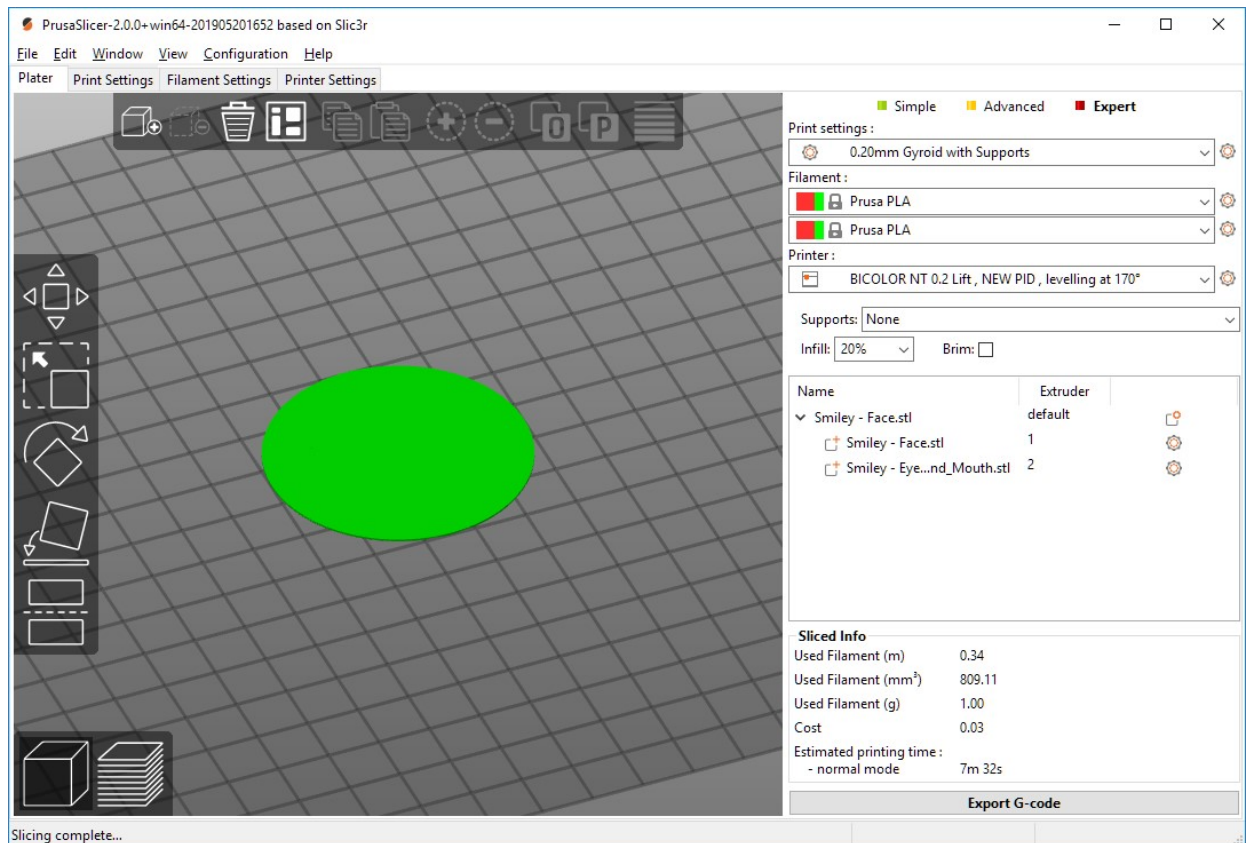
The best way to open multiple objects in PrusaSlicer is via drag&drop. Just select your STL files and drop them onto the main PrusaSlicer Window. It is important to know that PrusaSlicer will only detect multiple Stl's that you drag and drop onto it as Multicolor Object if you have a Multi-Color (Multi-Extruder) Printer configuration activated. Otherwise it will ALWAYS treat them as independent objects !! (Basically , make sure to follow step 2 and activate that configuration before continuing ☺). If your Object is detected as Multicolor, lucky you no alignment required... follow the procedure outlined in **Case 1** below . Otherwise you will need to manually align the components , this is covered in **Case 2** .

4) Case 1, Model is properly designed and aligned for multicolor printing.

This is clearly the easiest case, after dropping your STL's on the PrusaSlicer Main Window, you should see the following prompt :

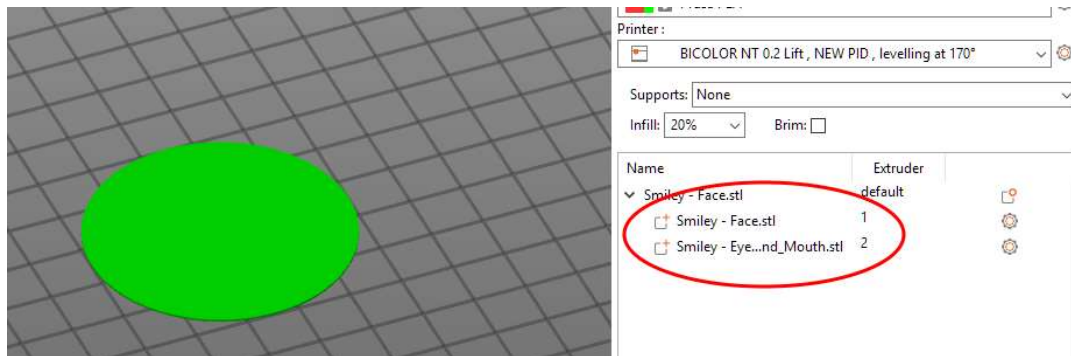


- Click on Yes , and you should see your model loaded . Don't worry if you miss the details and everything is in the same color, we'll take care of that now!

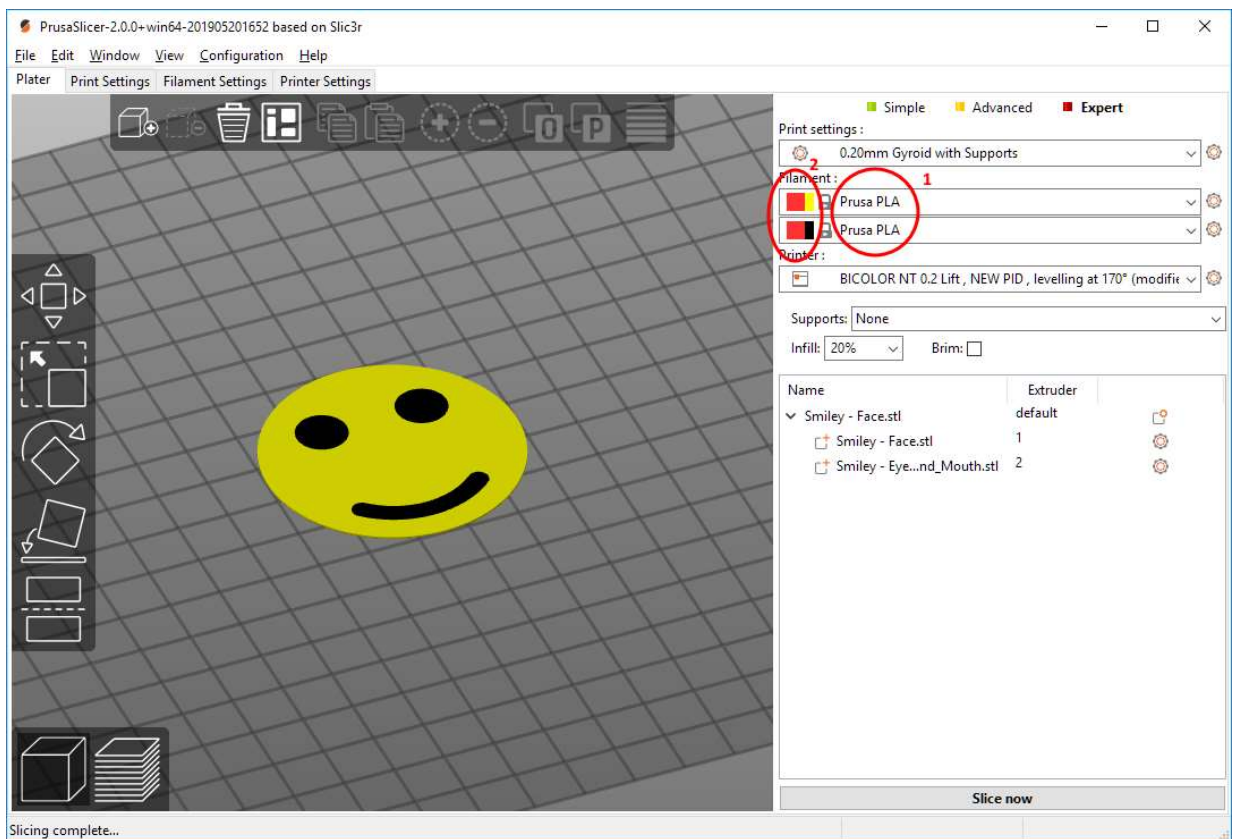


- Double-check your Extruder Setup in the window on the right side (Again , this window is only visible in Advanced or Expert mode , but you should be in Expert mode since the beginning of this tutorial anyway 😊)

In this example , the face will print on extruder (colour) **1** whereas the eyes and the mouth on extruder (colour) **2**



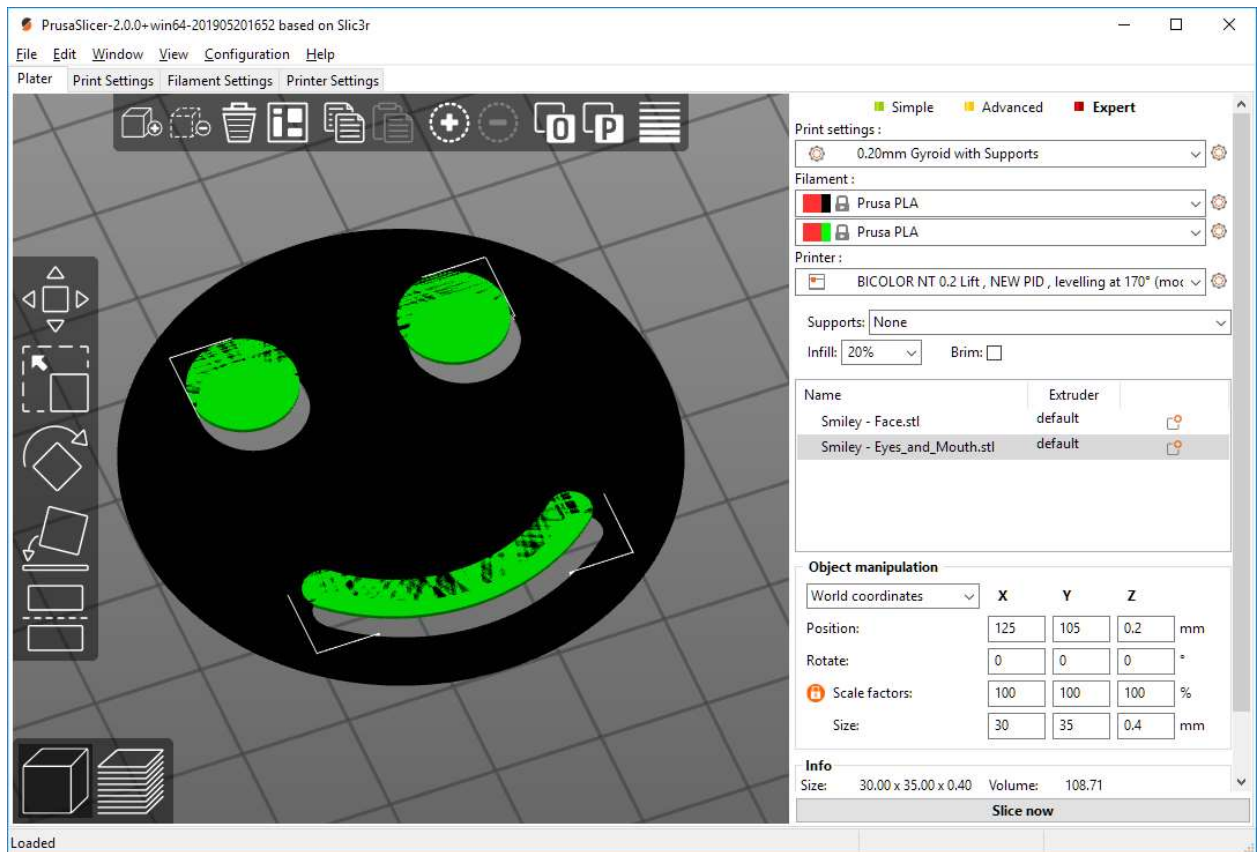
- In the main Window, select your Materials and choose colors by clicking on the left side of the colored box in front of the material name . Do this for all materials / colors required and you should end up with your PrusaSlicer looking like this :



That's it basically, your model is ready to be sliced, but please don't skip section 6 It contains important information that will help you understand how PrusaSlicer handles extruders and manages things ☺

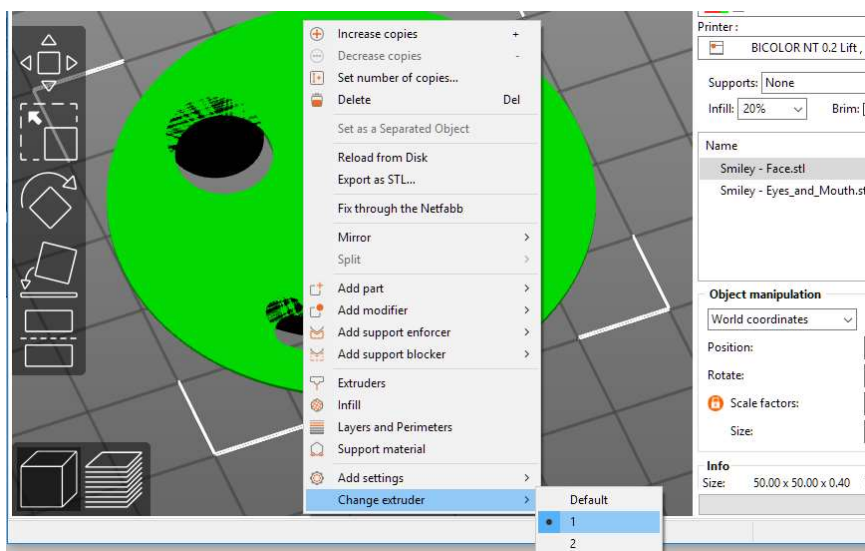
5) Case 2, Manual alignment is required.

If the parts have not been designed / aligned for multi-color printing, PrusaSlicer will load them as independent objects. That makes things a little more complicated as you have to align them carefully yourself. Your Window will probably look something like this after loading your objects :

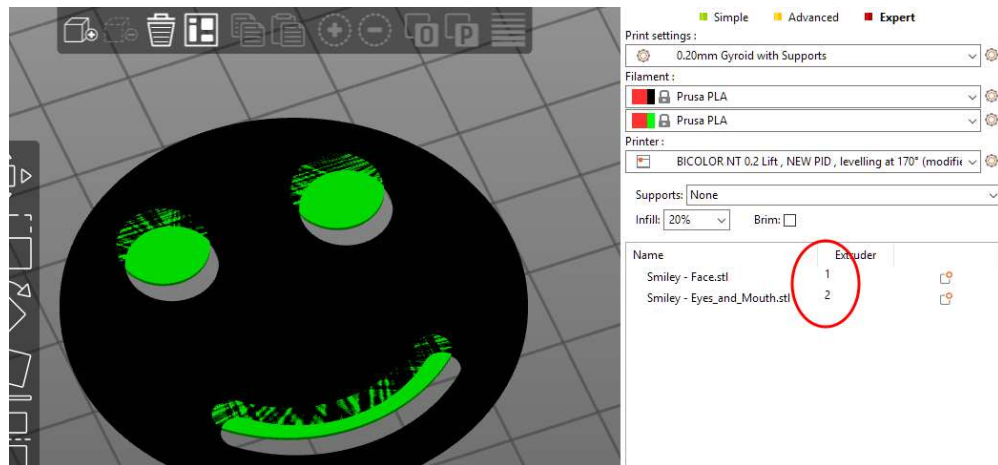


To make things even worse, both parts will have the default extruder selected , so that needs to be changed aswell !

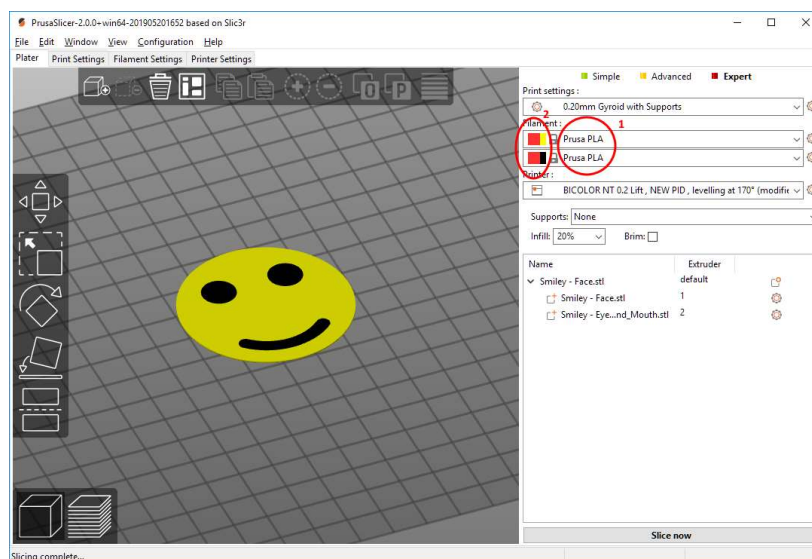
- In the Plater Window, right click on the part that you want to have printed first (The face in our example) , click on “change Extruder” , then select Extruder 1 .



- **ATTENTION** : You HAVE to click on the part in the plater window , right clicking the name of the part in the right window DOES NOT work !
- Repeat for all parts of your print , and you should end up with the extruders selected correctly :



- In the main PrusaSlicer Window, select your different materials on the top right and choose filament colors by clicking on the left side of the colored square in front of the materials(Colors are for the visual aspect only), then align the parts so they fit together nicely. Alignment can be a little tricky and is easier if you have a good contrast between the different colors used !



One thing to keep in mind is that you CANNOT align part on the Z Axis! If the color change is to happen on the top layer for instance, the STL's will need to be designed as true multi-color and imported as such in PrusaSlicer (procedure covered in Case 1)

That's it here as well, although the alignment can be a little tedious, your model is now ready to be sliced, but please don't skip section 6 It contains important information that will help you understand how PrusaSlicer handles extruders and manages things ☺

6) Things to bear in mind while selecting colors and the print order of the different colors:

(This is due to how PrusaSlicer organizes things)

- The Printer will ALWAYS start with EXTRUDER 1, then cycle thru them in numerical order
- The Printer will ALWAYS print the next FIRST color of one layer as FIRST COLOUR of the next layer, to minimize the number of color changes!!

Sounds complicated? Let's have an example based on this smiley

We have defined the eyes and mouth to be extruder one, so those will print first in the color we have selected (black) after that, the printer will move the head for the color change and we will load next filament (yellow). After the color change, the printer will then print the rest of the FIRST layer in yellow, immediately followed by the yellow part of the SECOND layer! After having printed the yellow part of the second layer, it will ask for the next color change, back to black. This model only has 2 layers, so it will end here, but otherwise, it will always print the LAST color of a layer as FIRST color of the following layer without asking anything!!!

- Darker colors will always tend to bleed into lighter colors , so always try to print Darker colors first , then go gradually lighter for best results
- Always make sure to extrude enough filament during the color change, the printer will ask you if the color is clear..... if you are unsure , answer no , and extrude a little more .
- For whatever reason unknown to me, the printer will extrude a little while moving back to the part after a color change. This is why I remove the material extruded during the color change at the very last moment (when the head is already lowering again) just by pulling on the end of it

7) Slice your model and start to print!

Yes, it is as easy as this, really ☺

If you are printing a lot of colors on multiple layers and are unsure about the order in which to load filaments, just open the G-code in your favorite Text editor and do a search for <M600>

```
3817 G1 X101.809 Y142.047 E0.01212
3818 G1 X102.586 Y142.823 E0.03447
3819 G1 X102.043 Y142.814 E0.01704
3820 G1 X101.599 Y142.371 E0.01969
3821 G1 X101.389 Y142.695 E0.01212
3822 G1 X101.673 Y142.978 E0.01256
3823 M600
3824 T1
3825 ; PURGING FINISHED
3826 G1 E-0.80000 F2100.00000
3827 G1 X108.967 Y78.805 F7200.000
3828 G1 E0.80000 F2100.00000
3829 G1 F2520
3830 G1 X110.606 Y79.095 E0.05221
3831 G1 X111.860 Y79.374 E0.04026
3832 G1 X113.376 Y79.769 E0.04912
3833 G1 X114.968 Y80.228 E0.05196
3834 G1 X116.448 Y80.700 E0.04870
```

The line just underneath will tell you what extruder comes next (which color to load).

T0 Is extruder1, T1 Is extruder2 etc.....

That way, you can write down all the changes you have to do in their correct order, and follow that list during the print.

8) Show me some examples of what can be done

- The smiley used for this write-up



- A custom cover for a Raspberry Pi case



- LCD cover and endcaps I used for my conversion to Full-Bear MK2.5



- A replacement box for the kid's UNO cards



~~~~~ The only limit is your imagination 😊 ( and your patience while changing colours ) ~~~~~

*I hope this small write-up has been useful and not overly complicated 😊 If you have comments or feedback, you can contact me via Thingiverse (Garfield1970) or email me at [garfield@vo.lu](mailto:garfield@vo.lu). Should you need help designing a part, you can try to contact me, I won't promise I can help, but if I have time, I can give it a try 😊 😊*