Slic3r settings

Slic3r is ‘slicing’ software that can turn a 3D model saved as STL file into a gcode file, containing instructions for the 3D printer. After starting slic3r a startup screen will appear (below). You can load your model by clicking “Add” on the top left of the screen. As soon as you load your model the software will begin slicing it.

The slicing software slic3r has separate ‘Print settings’, ‘Filament settings’, and ‘Printer settings’, which can be accessed on the top of the screen (below). Changing the print settings will cause the program to restart the slicing process. This is not the case with the Filament or Printer settings, provided you keep the same printer. Settings that you have changed can be saved, and saved sets can be selected on the right side of the screen (red box). There is a second choice for filament, because the makeit pro printer has two nozzles and can thus support two filament types.

The start screen (named Plater), shows the print bed, and your model, if you have loaded one. The colored lines on the print bed show where the nozzle will be centered.

![Slic3r start screen](image)

At the bottom of the screen, you can choose to view the 3D representation of your model, a 2D projection, and, after slicing has finished, a preview of the print and a layer view showing the path the printer will take on each layer.

In the rest of the document I'll show the settings I have used successfully.

Print settings

The print settings tab is subdivided in nine other tabs.

**Layers and Perimeters** sets general features of the layers, such as height. The printer makes a distinction between material on the outside (perimeter) and infill during printing. Increasing the number
of perimeters will give your print a thicker wall. I set this to five to have the cones on the petri dish consist entirely of perimeter, so they were printed in a single pass, rather than 2 passes (1 for perimeter, 1 for fill) possibly reducing stringing.
The **Infill** screen is used to set the degree and pattern of the fill. As the maximum thickness of the print was 2 mm, I chose a solid fill. The concentric pattern made the cones on the lid more smooth and stronger.
**Skirt and Brim** are terms for material printed before the actual print starts. A *skirt* is (usually) not connected to the print, but serves to start the printer and get rid of differently colored filament after a change. It also allows early troubleshooting before the print starts. Printing a skirt is always recommended.

A *brim* is a layer of material connected to the base of the print, increasing its footprint and reducing warping.
Support material is necessary when your print is wider than its base. This should be easily removable from the actual print, but will require some cleanup afterwards. As support material is typically brittle, it can cause problems during printing if hit by the nozzle.

Rafts are a basis for the print on the print bed, usually used to make the base of the print wider if necessary for stability, or make the print easier to remove from the bed. Using a raft on the ‘detachable’ setting caused my print to warp terribly.
The speed setting is key to a successful print, and slic3r allows to set different speeds for a range of aspects of the print. This is useful for prints with a lot of details, but not very applicable to mine. I have generally kept speed low (causing longer print times). For the cones I have modified the gcode file after slicing to print at a 10mm/s speed, and finish even at 5 mm/s.

Slic3r also has a setting under Filament > Cooling (see below) that affects the speed of the print for layers with really short print times.

I did not use the Mutiple extruders, Advanced, Output options, or Notes, with the exception of explicitly specifying the extrusion width as 0.4mm (nozzle size) under advanced.
**Filament settings**

The filament settings tab has two windows, **Filament & Cooling**. In the **filament** window you can specify the diameter of the filament and the temperature of extruders and the heated bed (if the printer has it). I did not specify the temperature of the heated bed here, but in the Printer settings > custom gcode (see below).

![Filament settings window](image)
The **cooling** window sets the options for the fan speed. Too much cooling will result in a brittle print. The amount of fanning required will depend on the hardware, but going 75% of max speed for most of the print (manually edited in the Gcode) has worked well for me.
Printer settings

The printer settings tab has four windows: General, Custom Gcode and one for each extruder.

In the general tab I have only specified the size of the bed (200 x 240) and origin (100, 120). The origin is probably wrong, but I have corrected for this using the custom gcode tab
The Custom Gcode tab provides three fields to specify actions in the Gcode: Start, End, and before each layer change. I have not used the latter. The start and end gcodes I have used can be copied into their respective boxes.

**My start Gcode**

```gcode
M117 MAKEiT Pro@Heating
M140 S60  ;Set target temperature for the heated bed.
M104 S[first_layer_temperature] ;Set target temperature for the nozzle 1.
M109 ;Wait until nozzle 1 have reached target temperature.
M190 ;Wait until heated bed have reached target temperature.
M117 MAKEiT Pro@Homing
G21 ;metric values
G90 ;absolute positioning
M82 ;set extruder to absolute mode
M107 ;start with the fan off
G28 SC ;Do homeing, clean nozzles © and let printer to know that printing started (S)
G92 X-100 Y100 ;center to bed
G92 E0 ;zero the extruded length
G1 F200 E3 ;extrude 3mm of feed stock
G92 E0 ;zero the extruded length again
G1 F3000
M117 MAKEiT Pro@Printing
```

**My end Gcode**

```gcode
M104 S0 ;extruder heater off
M140 S0 ;heated bed heater off (if you have it)
G91 ;relative positioning
G1 E-2 F9000 ;retract the filament a bit before lifting the nozzle
G1 Z+0.5 E-5 X+20 Y+20 F3000 ;move Z up a bit and retract filament even more
M117 MAKEiT Pro@Done
G28 X0 Y0 ;move X/Y to min endstops, so the head is out of the way
M84 ;steppers off
G90 ;absolute positioning
M81 ;Turns power off from the printer
```
The extruder tabs define the nozzle diameter and retraction. Nozzle diameter is fixed for a given printer. Retraction is pulling back the filament to prevent oozing and stringing during printing. Z-lifts tend to cause very thin web-like strings, so I do not recommend using this.