Slic3r Instructions

Created by Justin Nussbaum on 7/15/2015

jnussbaum@mail.usf.edu

File Preparation

- In order to load your part in the Repetier-Host program your part must be in the .STL file format. This can usually be done through the "save as" or "export" function of your CAD program.

Repetier-Host

- First load a part on the bed with the 'load' button.
- Under object placement you have the option of moving the object on the bed, rotating in any direction, scaling, copying or rotating your object.
- Before you can print you must slice your object into layers. You can modify how to slice your object under the slicer tab. To change parameters either choose one of the premade profiles (see how to make profiles below).
- Under the Slicer tab, you will have 3 options available for printing, 'Print Settings', 'Printer Settings' and 'Filament Settings'.
- You can modify these settings by clicking the configuration button above these buttons.
- After you have modified a setting click the save button and next to the name you can use a different name to save an additional profile. If the same name is left in the box you will automatically write over the previous profile.
- Please see below to see how each setting effects your print.

PRINT SETTINGS

*Hover your curser over an unknown parameter for an explanation.

LAYERS AND PERIMETERS
- Layer height: Layer height for each layer besides the first in millimeters.
- First layer height: Height of the first layer, keep this at a maximum value (0.3) for best first layer adhesion. This will keep your part from coming off the bed and help prevent warping on large parts.
- Vertical shells: Perimeters, aka shells on the Makerbots.
- Horizontal shells: Solid layers to generate at top and bottom surfaces: 2-3 is sufficient for thick extrusions (0.3 layer height), 3-4 is better for thin extrusions (<0.3).
- For best results leave quality boxes unchanged, and do not change advanced options.

INFILL

- Fill density: 0% = hollow, 100% = solid. If you need to make a solid part make sure to change *fill pattern* to one of the options listed under *top/bottom fill pattern*. See Figure 1 below.
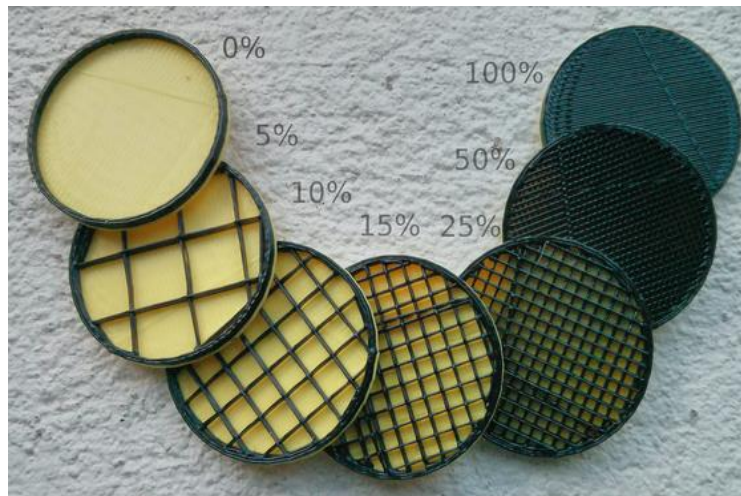

Figure 1: Various infill densities with rectilinear infill.

- Fill pattern: See Figure 2 below for the most common infills, an explanation of each and when each one should be used.

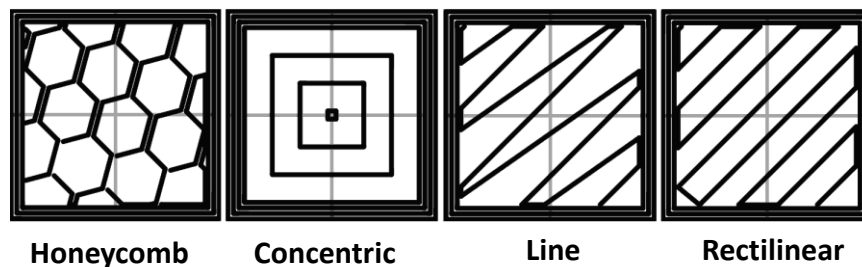| Honeycomb | Concentric | Line | Rectilinear |

Figure 2: Fill patterns. Honeycomb provides the highest strength to weight ratio and is the most common but will result in very high vibrations for dense infills. Concentric infill will outline the perimeters. Rectilinear is the second most common after honeycomb and should be used with high infill densities (or solid parts).

- Combine infill every: this option will combine infill layers. If you have thin layers, for example 0.1mm layer height, you can set this option to 2. It will then skip the first infill layer, and on the second layer create an infill layer 0.2mm thick and continue in this fashion for every other infill layer. This works well for reducing printing time and creating stronger parts. However, for parts with drastically sloped walls and/or very small layer heights and only a few perimeters there may be some issues. For predominately tall parts with not many slopes this option works well. The maximum value this can be set at is:
Current layer height * combine infill every # ≤ 0.3

Where 0.3 is the nozzle diameter.

- Solid infill every: This will make a solid infill layer every x number of layers.
- Fill angle: This option will rotate the infill x° every layer.
- Solid infill threshold area: When a layer's area is less than this amount it will make it solid. This reduces the likelihood of having weak spots in your parts. However, very thin high aspect ratio parts with solid infill sometimes suffer from nozzle drag and can be broken off.
- Only retract when crossing perimeters: Allows oozing of the nozzle inside the part. Thus any oozing will be invisible. Sometimes, too much retraction can jam the nozzle. It is best to leave this checked.
- Infill before perimeters: Leave this unchecked for best results.

SPEED

Be careful when increasing the speed settings. You can strip the filament with the hobbed (toothed) feed gear if set too high or cause the stepper motors to skip steps. You can tell if the printer is skipping steps when you hear it grinding (sounds like a loud buzzing noise) or your part looks shifted in one direction.

- I recommend following the default values for the speeds. Sometimes increasing speeds can decrease the quality of the part though, so you will have to find what speeds work best for your printer.
- Your infill speeds can be inaccurate and quick because they will be internal to your part and not very important for appearance.
- You want your perimeters to be slower to produce good quality surfaces.
- If you are having problems with adhesion to the bed lowering the first layer speed will dramatically help. Normal values for first layer speed range anywhere from 5%-40% or 10 to 30 mm/s for good adhesion.

SKIRT AND BRIM

- Loops: Number of times the part will be outlined before printing. This acts as a priming step for the nozzle. On the Makerbots the priming step is a straight line in front of the stage. 1 is sufficient for most object. For very small objects you may need 2. You just need enough loops so that the nozzle is fully extruding before the part begins printing.
- Distance from object: This is how far the loop will be from the object. 5 is a good default.
- Skirt height: How many layers thick the loops will be.
- Minimum extrusion length: Will create as many loops as necessary to extrude this set amount of length.
- Brim: This is the same thing as a skirt with 0 distance from object. It will create an outline around your object that will be attached around the base.

This works well to anchor parts to the bed that have a small contact area. It will remove easily after printing by hand. Occasionally a razor blade will be needed to clean up some of the edges.

SUPPORT MATERIAL

- If you want to generate support material the "generate support material" box must be checked.
- Overhand threshold: The angle from horizontal to enforce support material under. Leave this at 0 to automatically calculate where support material will be required. Setting at approximately 85 will enforce support material under most overhangs where setting it at 5 will enforce it only under flat surfaces that are not bridged (depending on whether or not 'don't support bridges' is checked, bottom option).
- Enforce support for the first x layers: This acts to support objects with a small contact area on the bed.
- Raft: This will create x number of raft layers.
- Pattern: Pillars or rectilinear are recommended for breakaway support material.
- Pattern spacing: Spacing between raft/support material lines.
- Pattern angle: Amount to rotate each raft/support material layer. If this value is set to anything above 0 it will be difficult to break away.
- Interface layers: Number of interface layers to create between the support and the part.
- Interface pattern spacing: Spacing for the interface between the support material and the part.
- Don't support bridges: Whether or not to create support material under bridges. A bridge is a point between two connecting pillars or points. A cantilever is an example of a point that is not bridged.

MULTIPLE EXTRUDERS
- Choose which extruder you want to use for each of the listed options
- Ooze prevention: Enable to help prevent the second nozzle from oozing when not being used.

NOTES, OUTPUT OPTIONS – N/A

ADVANCED

- Default extrusion: Width of the default extrusion
- First layer: Leave this around 200% (150%-250%) for best adhesion to the bed. This will prevent the part from coming off during printing and warping on large parts.
- Perimeters: 90% will produce more accurate smoother perimeters.
- Infill: Extrusion width of infill, increase this to around 200% for solid parts to speed up printing speed. Leave at 0 for all other parts. 200% is also helpful for parts with layer heights <0.10mm
- Solid Infill: Leave at 0 for best results.

- Top solid infill: Leave at 90% for best results, this will produce a smooth top surface.
- Support material: Leave at 0 for best results.
- Bridge flow ratio: This option will extrude less than required when bridging. This acts to tension the depositing filament to reduce sag. 0.8-0.95 works well.
- Threads: Set this to 1 above the number of cores/processors your computer has. If you're not sure how many processors you have, set this value to 3.
- Resolution: Leave this at 0 for best results. For high resolution parts this can be increased above 0 to reduce slicing time.

FILAMENT SETTINGS

FILAMENT
- Diameter: Filament diameter. Measured by a caliper in multiple places and averaged.
- Extrusion multiplier: Should only be set at 1, if there is something wrong with the extrusion, the diameter was probably measured wrong, or if there is an issue with the first layer, the height is probably incorrect. See Z-offset option in printer settings.
- Temperatures: After you determine what the optimum temperature is set this value for *other layers*. Unless adhesion to the bed is an issue, put your *first layer* temperature 5°C above your other layers temperature. If your part is warping off the bed or peeling off the bed you can set this value at 10°C or above. If more than 10°C above, test the *first layer* temperature first though to make sure this temperature does not burn the filament.
- Default first layer temperature for PLA is 60°C. This can be increased to 80°C if poor adhesion is noticed. Set other layers to 60°C. For ABS the first layer should be between 100°C to 110°C for both the first layer and other layers.

COOLING
Do not modify these settings.

PRINTER SETTINGS

GENERAL

- Bed size: 200mm for X and Y
- Origin: 0, 0
- Z Offset: This changes the height that the nozzle will print the first layer. This is VERY important to get a proper print. Setting this at a negative value will move the nozzle down from the homed position, while setting it positive will move it in a positive position from the homed position prior to printing. In order to make sure you have a proper layer height the solid deposition lines

(perimeters and infill on your first layer) should be just barely touching, NOT overlapping. They should have a smooth top surface. If you have a thick layer height (>0.2) if this is not perfect it will be ok and made up in subsequent layers. If you have very thin layers (<0.2) and you don't have a near perfect first layer you may end up with a failed part.
- DO NOT CHANGE ANY OTHER OPTIONS IN THIS SCREEN.
- Set extruders to two.

CUSTOM G CODE
- The custom G-Code will move the nozzle up before printing to allow any nozzle ooze to be removed before the print begins. The nozzle ooze should be removed before the print starts.

EXTRUDER 1/2

- Nozzle diameter: 0.3 for these printers
- Offset: Measure the distance between your nozzle tips with a set of calipers. This offset will be input here. You may have to make some fine adjustments to this. If your filament from nozzle 2 is overlapping the filament from nozzle one this number needs to be decreased slightly. If your filament from nozzle 2 has separation between the filament deposited from nozzle 1 then you need to increase this number slightly.
- Retraction length: Amount of filament to retract when a retraction is trigger. Setting this to a value which is too high can result in the hobbed bolt stripping the filament. Depending on the retraction speed values between 2.5-3.5 is normal. If the retraction speed is reduced then the retraction distance can be increased.
- Lift Z: If this value is greater than 0, then the nozzle will lift off the part in addition to a retraction. Then move back down after a large move. A large move is defined by the length "Minimum travel after retraction". Usually this is not required and can lead to longer print times when enabled.
- Extra length on restart: Leave at 0 for best results.
- Minimum travel after retraction: Only trigger a retraction for non-print movements whose distance is greater than this value.
- Retraction when tool is disabled: Length = 7+, increase if second nozzle keeps oozing when it is not being used.
- Leave the rest of the options on this page unchanged.

GOOD LUCK!