

REPETIER-HOST

Jonathan Keep

Repetier-Host is a free software from Germany that will slice a 3D model saved as a .stl file into gcode, the code that the 3D printers work from. Repetier-Host is basically a user friendly 'front window' that hosts two gcode slicer programs that are Slic3r and Skeinforge. Slicer3r is the easier to setup and this document covers how to set this up to work with paste materials, specifically clay on a Make your own Delta Ceramic 3D Printer.

The program can be downloaded from [here](#) and is available for most operating systems.

Have a look at the [Table of content](#) at the bottom of the page.

There is a [Repetier Wiki](#) that is helpful.

There is also a [Troubleshooting](#) page.

FIRST SETUP PRINTER SETTINGS

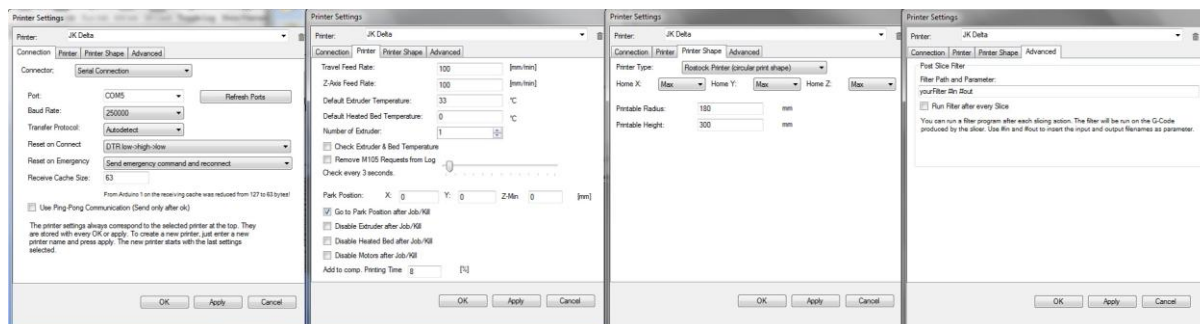
Printer settings, under 'Config' dropdown (third across from top left) once you open Repetier-Host, opens a new window that has four tabs that need looking. Above in the top dropdown set a name for your printer settings replacing default. Enlarge this PDF and you should be able to read the settings on the image below.

Tab 1 – Connection, I do not connect Repetier-Host to my printer so do not set this. (The computer next to clay and the printer is not a good idea)

Tab 2 – Printer, I think most of this gets over written by the Slicer program but these are my settings.

Tab 3 – Printer Shape, this is important as it gives a visualisation of your printer volume in the main program window. Ensure you select circular/delta printer in the drop down. All 'homing' I set to max and radius 180 mm and print height 300 mm. The apple version I think looks a bit different and I have had trouble setting this so you might have to fiddle a little - I am using windows.

Tab 4 – Advanced, I ignore.



NEXT REPETER-HOST MAIN WINDOW

After setting printer settings in Config I set nothing else in the main window. I move over the right hand side where in the main window there are a further four tabs.

Tab 1 - Object Placement, here I click on the + graphic (top right) and this is where you drop a .stl file into the host for slicing. If your file is well prepared nothing further needs to be done but there are scaling, rotating options. The 'Centre Object' button just along from the + button can be useful. When I want to slice the next file I use the – button to remove the model and then + to add a new .stl file.

Tab 2 – Slice, once the model is placed in tab 1 move onto tab 2 to slice it. Here you can chose (click on Active) to either Slice with Slic3r or Skeinforge. Each need to be setup with a profile relevant to the material you are printing with. Below is the setup for Slic3r and most of the time works really well. Skeinforge is a more complex slicing program with a lot more setting parameters and is worth a look if you have time, and patience.

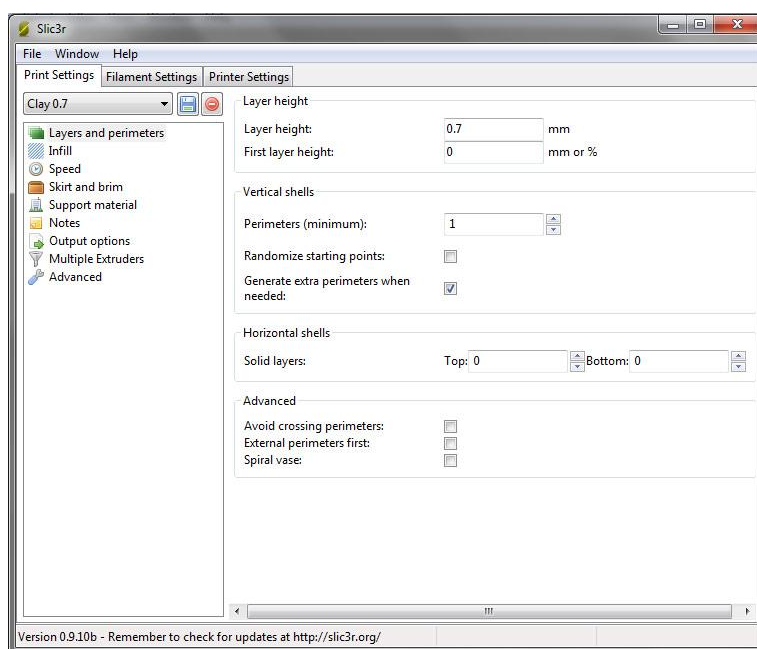
Tab 3 – G-Code Editor, once the slicing has completed the Repetier-Host main window automatically moves onto this tab to show the gcode that is what the printer works from. All being well in this tab all you need to do is save the gcode to your computer or directly onto the SD card that is slotted into the printer. You save by clicking on the floppy disk emblem top left in the G-Code Editor tab window. Once saved, back to Object Placement (tab 1) to slice your next model.

Tab 4 – Manual Control, I do not get involved with but if you do you will need to set up the connection in Config/Printer Settings, top left in the main window. Connection is with a USB cable from your computer into the Arduino of the printer.

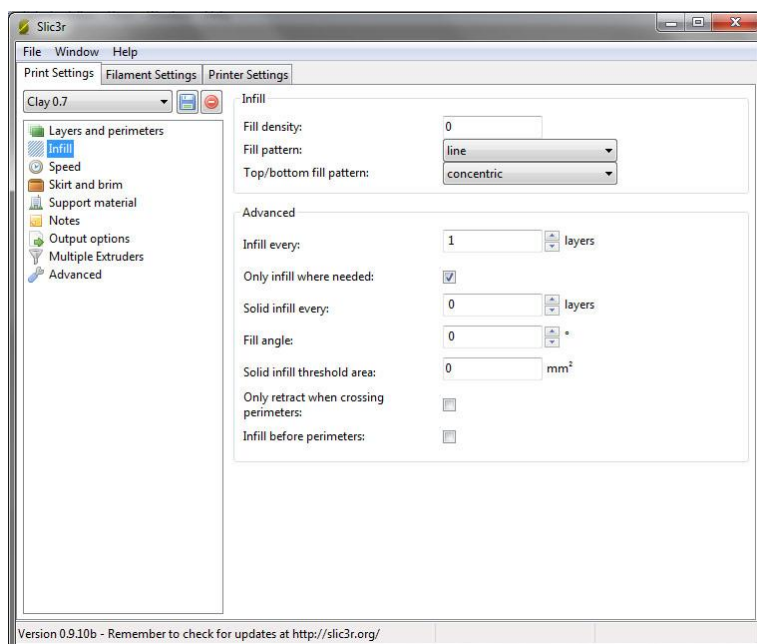
FINALLY SETUP SLIC3R

In Repetier-Host main window go to Slicer tab (upper right). Click on the Configure button for Slic3r, upper right. A new window opens that is Slic3r. This has three tabs that you will notice correspond with drop down menus back in the main slicer window under Slic3r. They are Print Settings, Filament Settings and Printer Settings. You will set a profile for each of these in the smaller Slic3r window that has opened. Once set the window is closed and you return to the main window and find the setting name in each of the drop downs under Slice with Slic3R, Print Settings, Printer Settings and Filament Settings. Once this configuration is done you are ready to slice files as above.

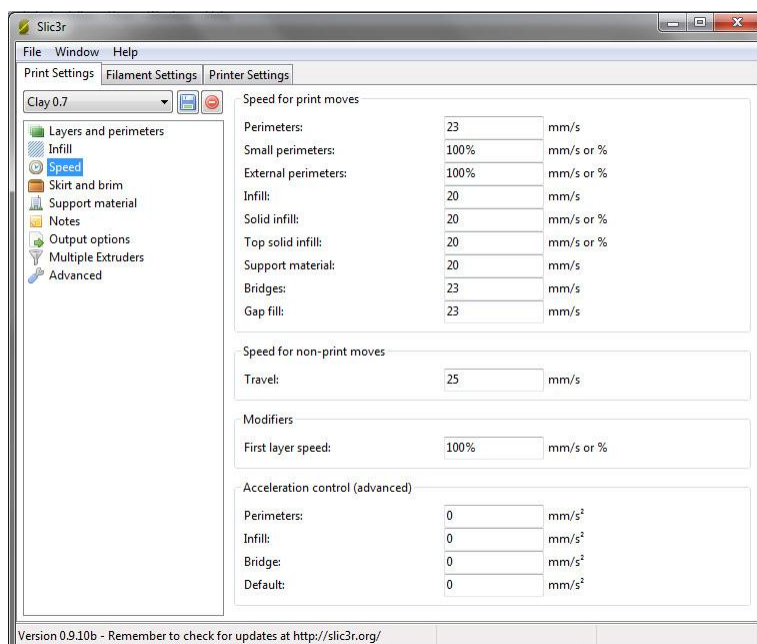
Tab 1 - **Printer Settings**, this tab has 9 subsections (left column) all pictured below. After making a first setting change the profile name will say (modified). Click on the floppy disk/save icon and give your profile a new name and select OK. There after each time you modify you must save and just OK the same profile name until you want to set up a new profile. Selecting a set up profile can become a copied template for a new profile but give it a new name. Print settings below. Zoom in PDF to read settings. Hover your mouse over any setting box to get a brief description.



Layers and perimeters, this is the slice height, 0.7 is average for clay. For larger scale I go up to 0.9 and smaller scale with a smaller extruder nozzle I will go down to 0.6. Vertical shells is the number of walls to the print. I work on 1 for speed but for stability or thickness you can go 2 or 3.



Infill, making hollow vessels I use 0 fill density that makes all other settings inactive. This is a setting to play with if you want internal supporting structure.



Speed I tend to print at 23 mm per second. The travel speed is when the printer is moving up and down to the home position.

(No picture)

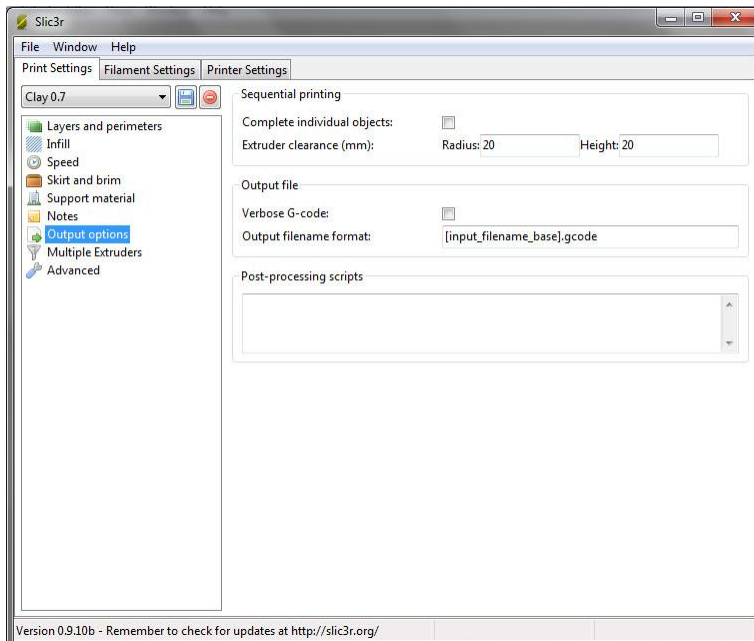
Skirt and brim, is used to help fix plastic prints to the print bed and not needed with clay so I set everything to 0.

(No picture)

Support material, again with clay I am not using so the tick box is not selected and all else is set to 0.

(No picture)

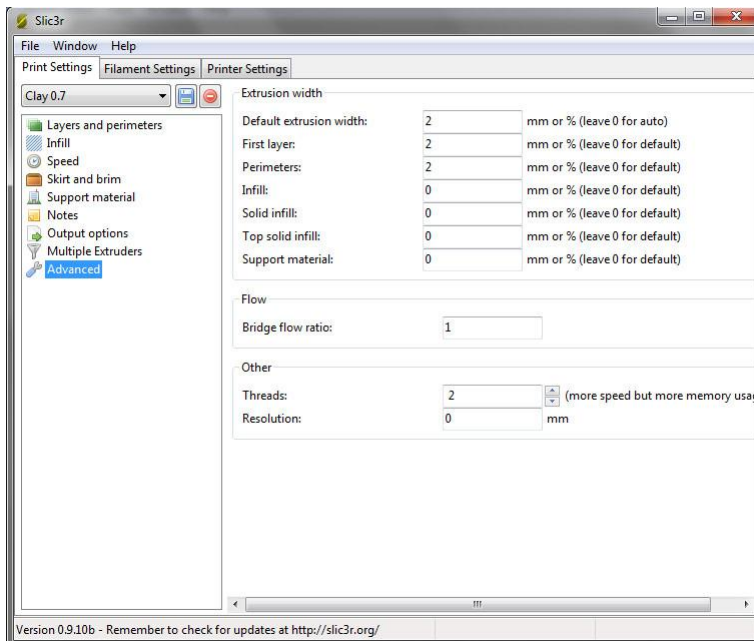
Notes, no notes, left blank.



Output options, basically nothing selected but the output file format must be .gcode.

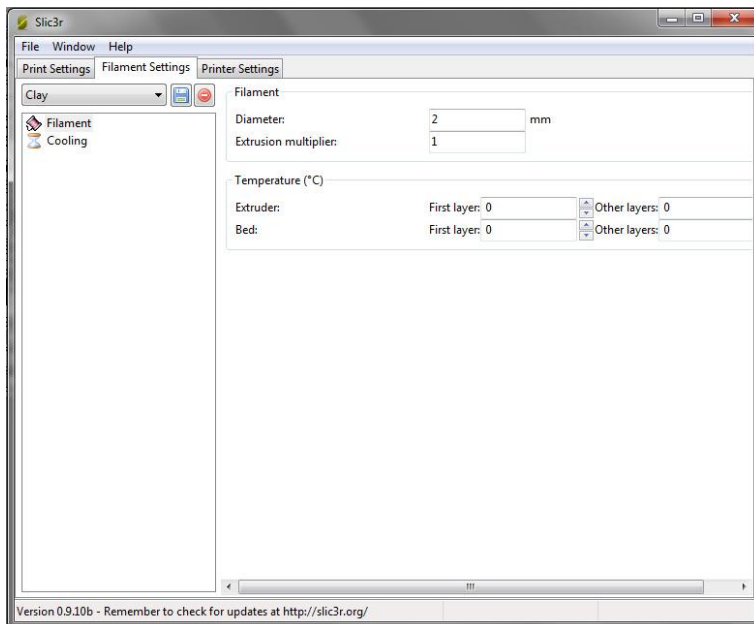
(No picture)

Multiple Extruders, there is only one so all set to 1.



Advanced, extrusion width 2 mm but for larger scale could be 3 mm.

Tab 2 – **Filament Settings**, again set up a profile name as back in the main window you will need to select this in the slicer Filament Setting box.

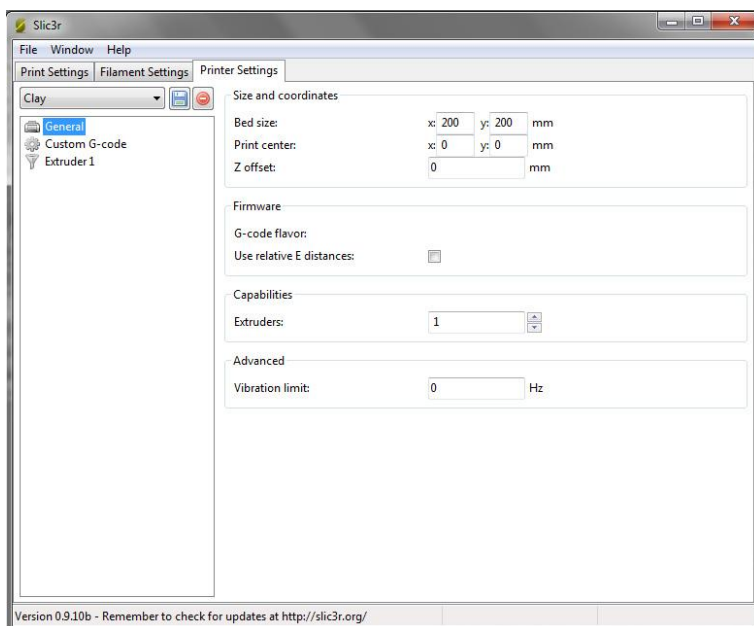


Filament, I work on 2 mm.

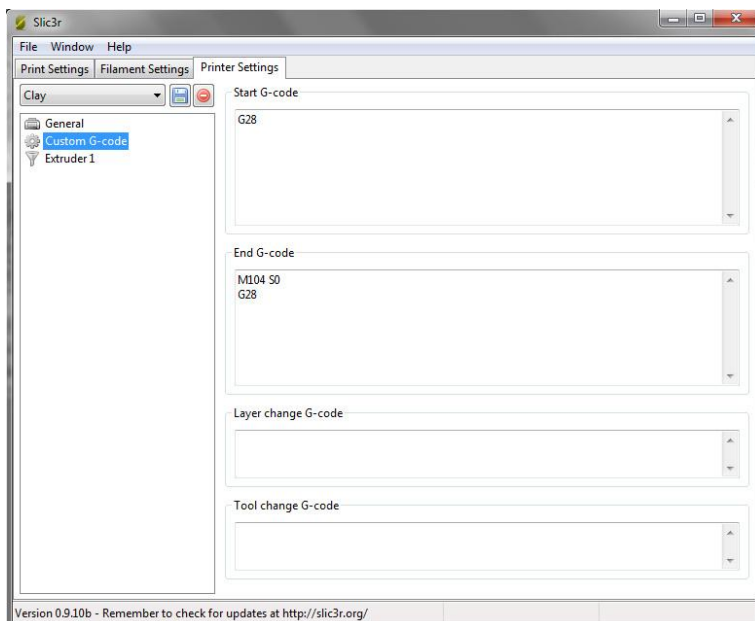
(No picture)

Cooling, there is no cooling so noting is selected under Enable.

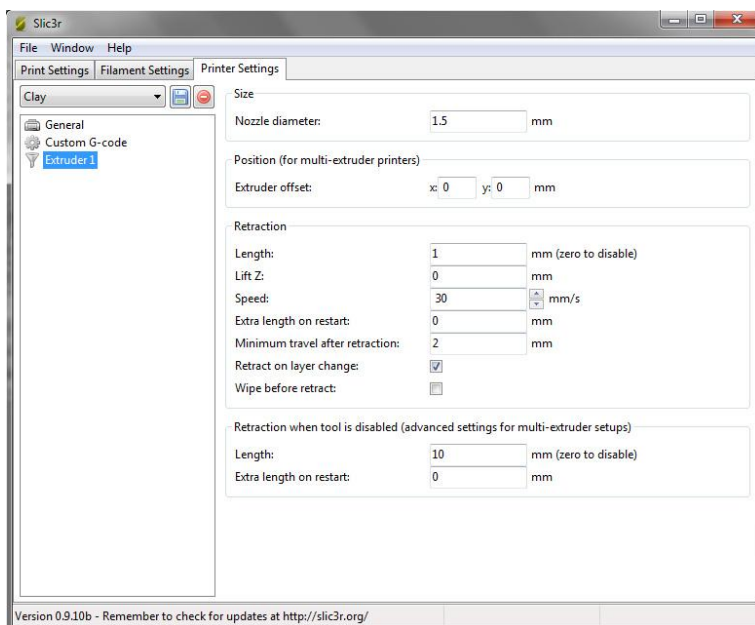
Tab 3 – **Printer Settings**, set up a profile name as you will need to select this back in the main window from the drop downs.



General, relatively strait forward.



Custom G-code, if I remember correctly this is the default. You do want G28 in the start and end G-code. G28 is the command to go to home position that is at the endstops at the top of the printer.



Extruder 1, the settings for the clay print head.

This YouTube video covers working with Repetier-Host -

http://www.youtube.com/watch?v=R5Cg8ioxlrg&feature=c4-overview&list=UUvFXo_8dqYWhdq7bQJLh5A

For further information on ceramic 3D printing see - http://www.keep-art.co.uk/journal_14.html