Slicer

07 November 2019   15:57

- When you first install Slic3r, it will go through a basic setup to set some of your basic parameters for your printer.
- Opening a model file, you will see the model and on the right, you'll see the THREE configuration files that together setup your print (figure 1). They will all be on My Settings. This is your starting point.
- If you click on the little cog-wheel next to the particular settings file, it will open a new tab showing those settings
- I would suggest that it is worth getting a baseline configuration working that is under "My Settings"
- If you make any changes to one of the settings files, then the save button is on that setting's tab (figure two)
- Once you want to start moving on from the base My Settings you can export your config
- IMPORTANT, THIS WILL HAVE YOU OVER!: If you load a new config file AND your "Print Settings"/Filament Settings"/"Printer Settings" tabs are still open from the previous config, you need to close those tabs and then re open them from the Plater tab (figure 1).
- It's not clear what the difference is between Export Config and Export Config Bundle. The manual isn't clear either, and what "I" understand from it, isn't actual behaviour of the software (both seem to export everything). But it would seem prudent to do an Export Config Bundle as that clearly exports everything to one file.
- When you load a config file, make sure you select the correct file from the drop down list next to "Print Settings"/Filament Settings"/"Printer Settings" in the Plater tab
- On the Plater tab, press Export to G-code, to slice the model and create the relevant file for your printer
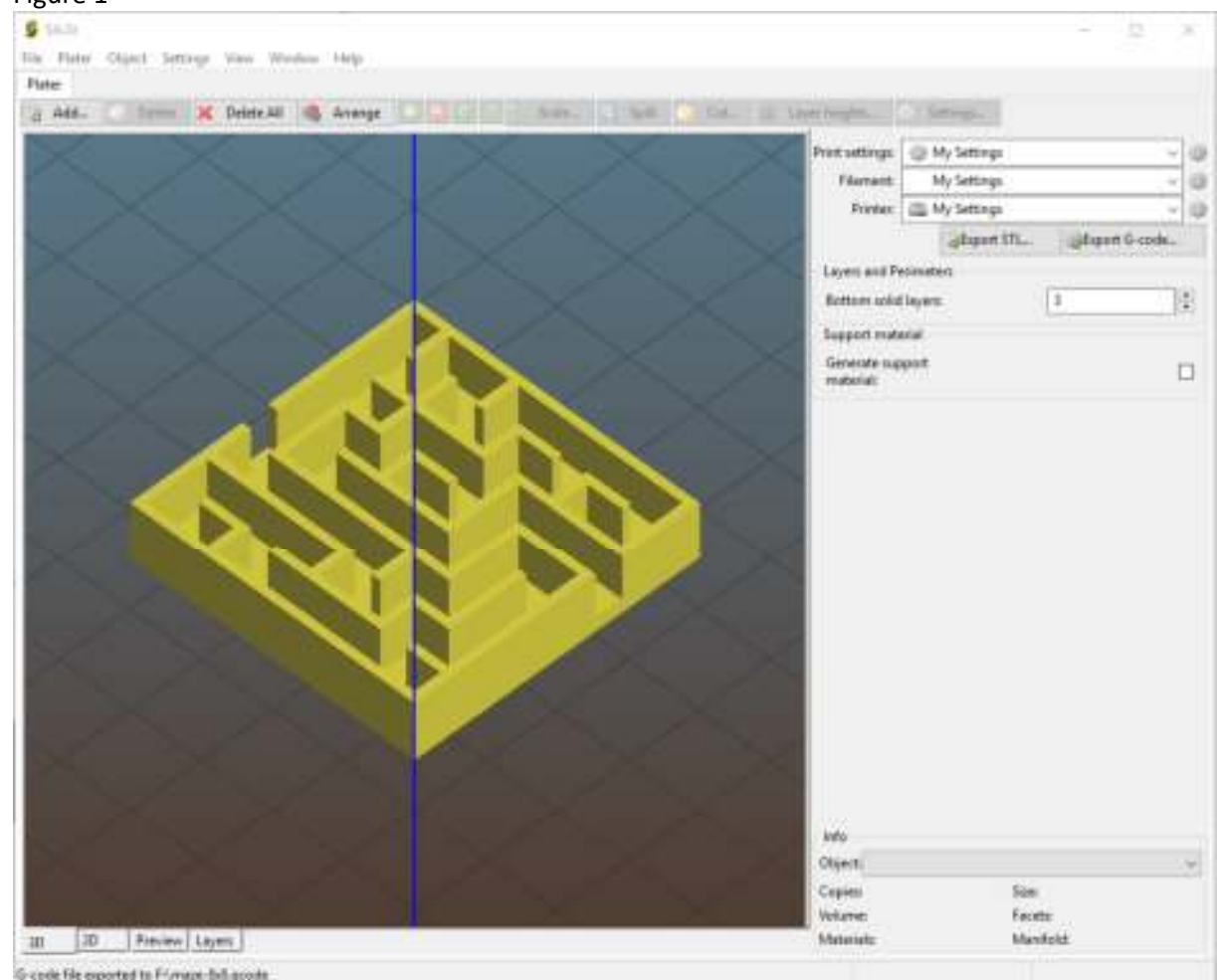
Figure 1

Figure 2: The save button!

Slic3r

File Plater Object Settings View Window Help

Plater Print Settings ✕ Filament Settings Printer Settings

My Settings

Layers and perimeters	Layer height		
Infill	Layer height:	0.2	mm
Skirt and brim	First layer height:	0.3	mm or %
Support material	Use adaptive slicing:	☐	