Prusa Slicer

Prusa Slicer

Prusa Slicer is the version of 3d Slic3r adapted for the Prusa i3 MK3S

Prepping your 3d model for printing

1	Open the **PrusaSlicer** application. If you are prompted that a Configuration Update is available, click to install the update before proceeding further.
2	Use the **Add** button to load models into PrusaSlicer
3	**Delete** **Delete All** buttons remove the model(s) from PrusaSlicer opens the detailed settings of print , filament and printer
4	Use **Move**, **scale**, **rotate** , **Place on Face** and **cut tools** to prepare the model to printing in the most efficient/ effective orientation,
5	Select a **Quality / Speed** setting for your print
6	Material selection (on Fabrication Lab computers, only filament brands/polymers provided in the Lab will appear in the dropdown list)
7	Select a **Supports** setting
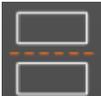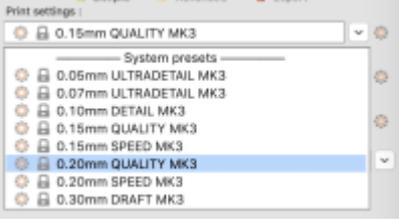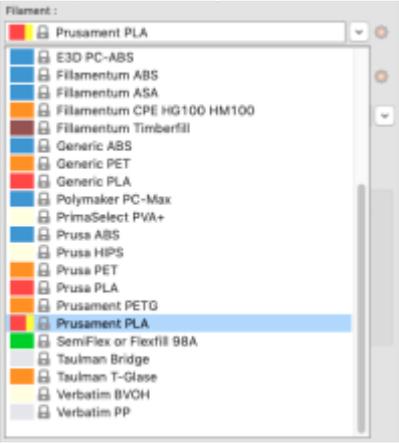
8	Select an **Infill** percentage
9	Click **Slice** [Slice now] to *slice* the model into layers in the Z axis and select **layers preview** to view infill land
10	Check the **Sliced Info** for the cost and how long its going to take to print before exporting your gcode *The Edge charges 15cents / gram and your print needs to be finished before the Lab Closes - talk to the Fab Lab Supervisor if you have a long job you'd like to do .*
11	Generate and export *gcode* to the SD Card by clicking the **Export G-Code** button [Export G-code].
Hint	Right-click on model opens a context menu
Hint	Switch between 3D editor and Layer Preview by clicking &