












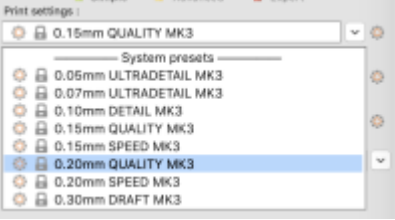
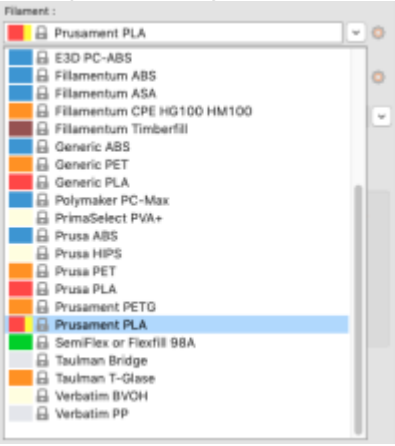

Prusa Slicer

SLQ Wiki Fabrication Lab 2024/10/02 06:39

Prusa Slicer

Prusa Slicer is the version of 3d Slic3r adapted for the [Prusa i3 MK3S](#)

Prepping your 3d model for printing

1	<p>Open the PrusaSlicer  application. If you are prompted that a Configuration Update is available, click to install the update before proceeding further.</p>
2	<p>Use the Add  button to load models into PrusaSlicer</p>
3	<p>Delete  Delete All  buttons remove the model(s) from PrusaSlicer opens the detailed settings of print , filament and printer</p>
4	<p>Use Move,  scale,  rotate , Place on Face  and cut tools  to prepare the model to printing in the most efficient/ effective orientation,</p>
5	<p>Select a Quality / Speed setting for your print </p>
6	<p>Material selection (on Fabrication Lab computers, only filament brands/polymers provided in the Lab will appear in the dropdown list) </p>
7	<p>Select a Supports setting </p>

8



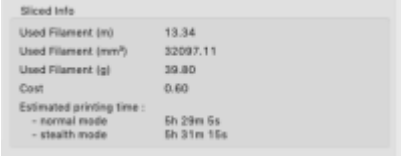
Select an **Infill** percentage

9

Click **Slice**  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



Hint Right-click on model opens a context menu

Hint

Switch between 3D editor and Layer Preview by clicking  & 