












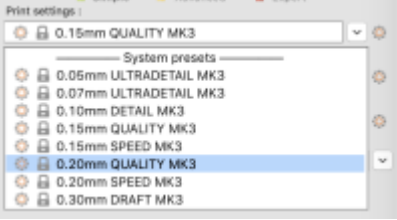
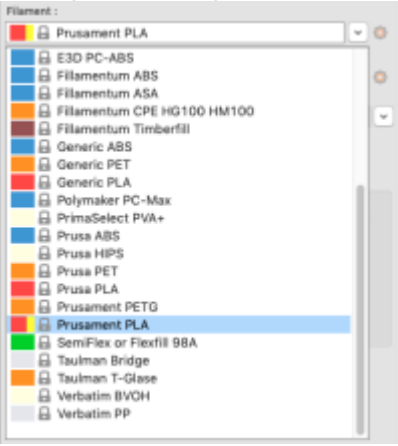

# Prusa Slicer

SLQ Wiki Fabrication Lab 2026/05/30 20:48

# 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 <b>PrusaSlicer</b> application. If you are prompted that a Configuration Update is available, click to install the update before proceeding further.</p>
2	 <p>Use the <b>Add</b> button to load models into PrusaSlicer</p>
3	  <p><b>Delete</b> buttons remove the model(s) from PrusaSlicer opens the detailed settings of print , filament and printer</p>
4	     <p>Use <b>Move</b>, <b>scale</b>, <b>rotate</b>, <b>Place on Face</b> and <b>cut tools</b> to prepare the model to printing in the most efficient/ effective orientation,</p>
5	 <p>Select a <b>Quality / Speed</b> 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 <b>Supports</b> setting</p>

8



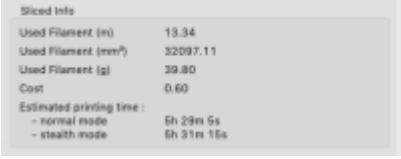
Select an **Infill** percentage

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  & 