









Prusa Slicer

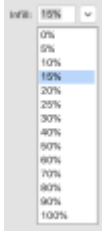


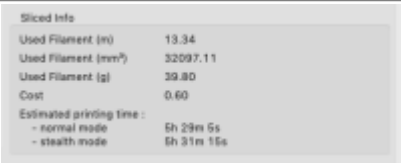
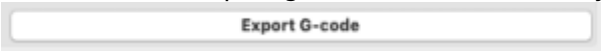

SLQ Wiki Fabrication Lab 2024/04/26 10:43

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, 
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 
7	Select a Supports setting 

8	 Select an Infill percentage
9	Click Slice  to <i>slice</i> 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 <i>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 .</i>
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  & 