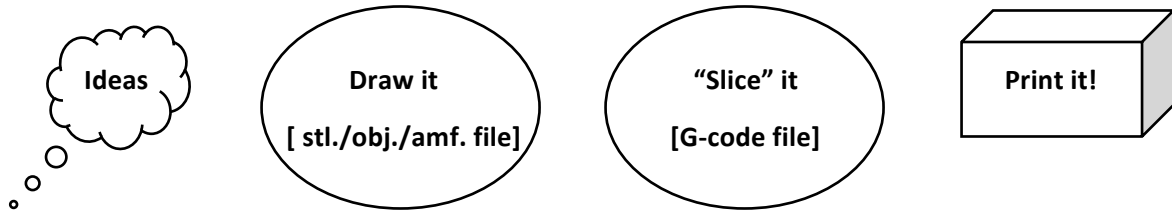


3D Printing

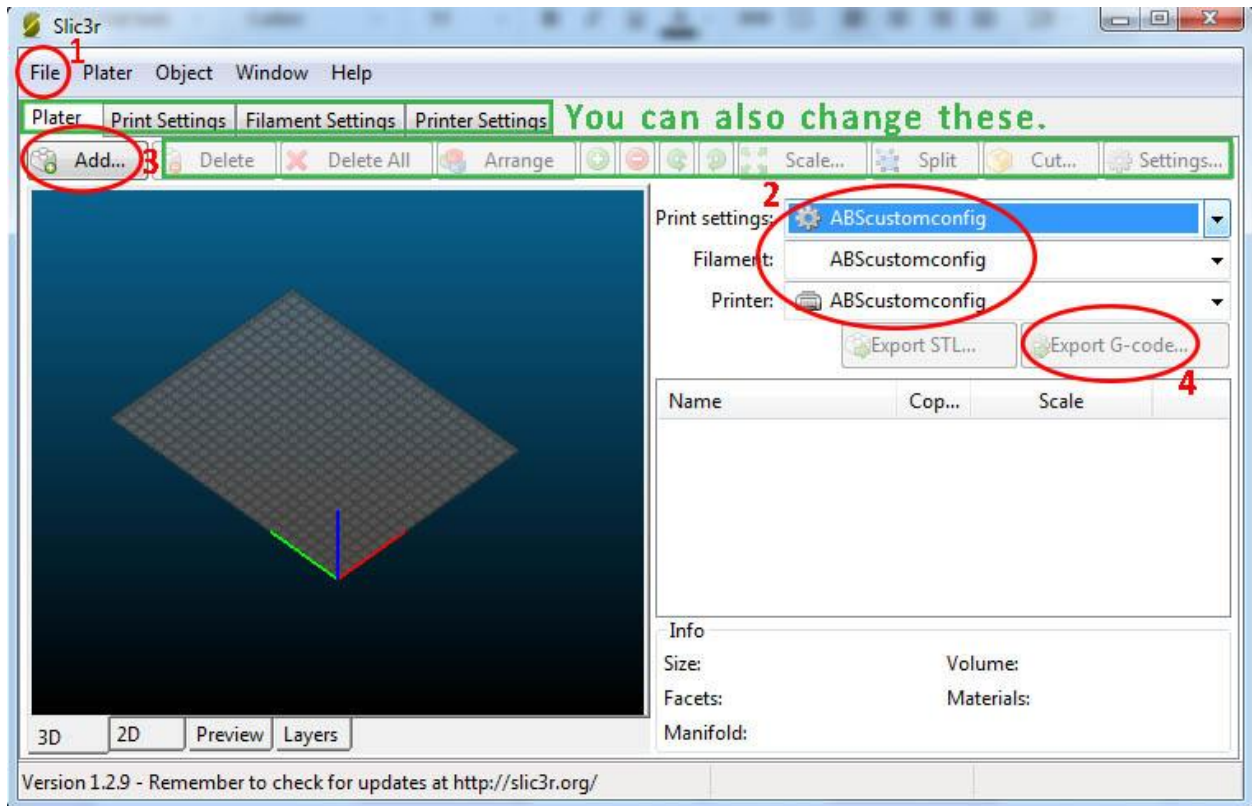
JLEE 20150805

4 Simple Steps!



If you have a stl./obj./amf. file:

- 1) Download Slic3r from <http://dl.slic3r.org/win/>
- 2) Download Slic3r_config_bundle.ini from <http://openlab.ece.illinois.edu/>.



- 1) Go to File>>Load config_bundle file, load the Slic3r_config_bundle.ini you just downloaded.
- 2) Select either ABScustomconfig / PLACustomconfig for all 3 Print settings, Filament, Printer depending on which filament(ABS/PLA) you are using.
- 3) “Add” your stl./obj./amf. file.
 - a) Refer to the section “You can also change these (Slic3r)” below.
- 4) Press “Export G-code.”

5) Refer to section “If you already have a g-code file” below.

You can also change these (Slic3r):

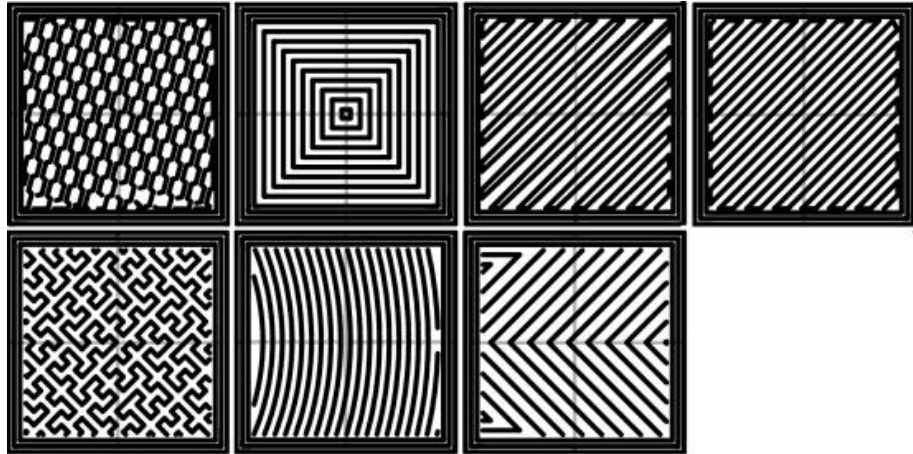
Go to “File” >> “Preferences” >> Mode: “Expert”

1) Page: Plater

- a. Arrange and/or scale your object. (You can check size & volume of your object!)
- b. Right click on your object to flip/rotate it.
- c. Add more objects to print at one go. Arrange them if necessary.

2) Page: Print Settings

- a. Infill
 - i. Layers and Perimeters
 1. Decrease layer height if you want a more beautiful end product.
 2. Adjust vertical & horizontal shells perimeters if necessary.
 - ii. Infill pattern & density(reduce density to reduce filament used if structural strength isn't important)



Infill patterns at 60% – top left to bottom right: Honeycomb, Concentric, Line, Rectilinear, Hilbert Curve, Archimedean Chords, Octagram Spiral

- b. Skirt(the perimeter around your object- to make sure filament will come up smoothly prior to printing object) & Brim(make the first layers slightly larger, useful to reduce warping on object) & Raft (printing a few layers of support material under the print)
- c. Support material – Generate this if your object isn't 3D Printer friendly.
- d. Speed - Decrease speed for small perimeters if you're printing an intricate object.

2) Page: Filament Settings

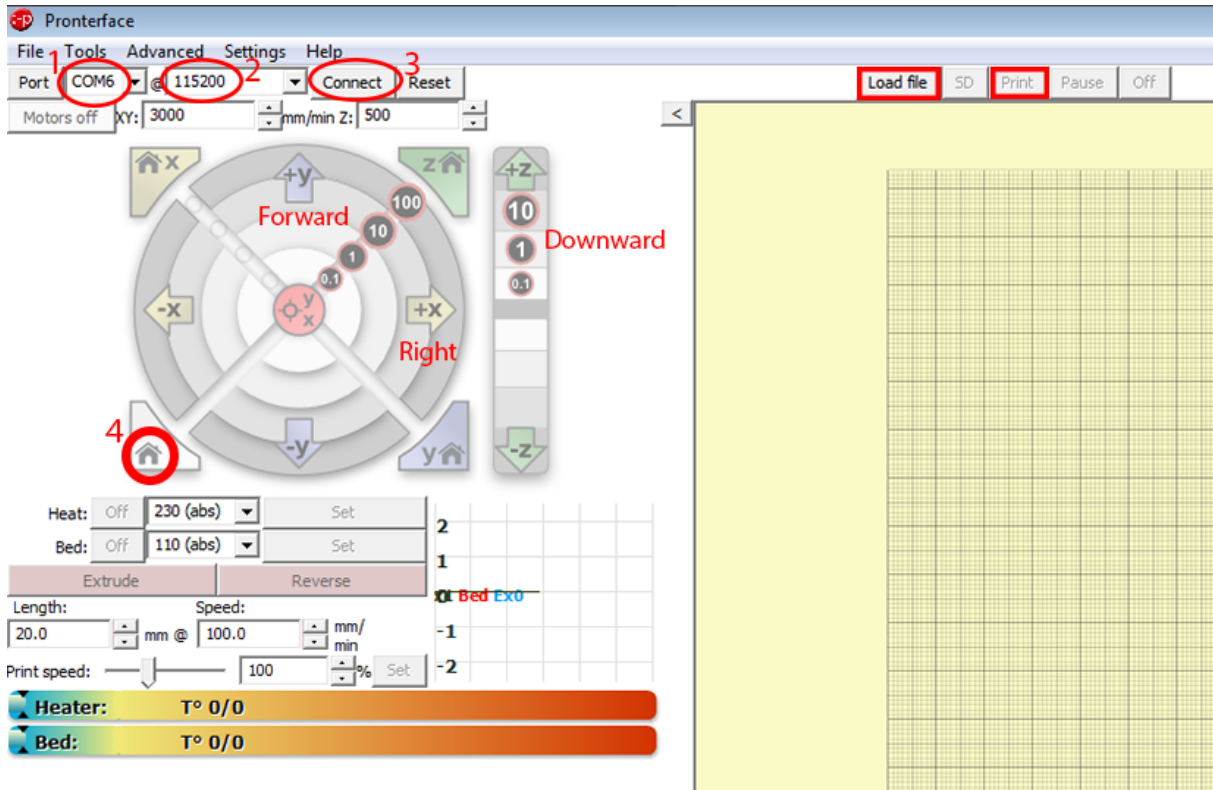
- a. Cooling - Can select “Keep fan always on” if you're printing an intricate object.

3) Page: Printer Settings

- a. General – Change Z offset if the nozzle is too near/far off the print bed.

If you already have a g-code file:

Download Pronterface: <http://koti.kapsi.fi/~kliment/printrun/> (There is an older version Slic3r in this bundle.)



Set up

- 1) Port: COM6
- 2) Baud rate: 115200
- 3) Press "Connect"!
- 4) Skip to "Print" section below to start printing OR to "More" to play around with Pronterface & the 3D printer.

More (Optional)

- 1) Press "HOME" - MUST DO (Note: Don't travel beyond the initial position for all axes! To be safe, travel a small step at a time. Or press the "Reset" button to kill operation.)
- 2) Calibrate -> Travel around all four corners, adjust 3 screw on the bottom of the heat bed until the distance between the nozzle and the heat bed is a paper's thickness (move a paper in between that gap, the paper should be movable but with some friction).
- 3) You can manually heat the hot end/heat bed/extrude filament using Pronterface too (You need to do that to change filament).
- 4) In case of emergency: press "Reset" button / turn off the power supply.

Print

- 1) Press "Load File" , then "Print"
- 2) Wait..... (The print bed and hot end are being heated up.)
 - a. Read the command line window to see how long it takes to print your object.
 - b. Don't let the computer sleep.
- 3) If interface is locked during a print, press 'unlock' at the top right corner.
- 4) **Be Careful!** : Print bed & Nozzle will be hot prior to print until they are cooled down after the print. Nozzle will reach ~464 Degrees Fahrenheit during print.
- 5) Wait for the part(s) to cool down, then remove it.

If you have no idea what to print:

Find stuffs from <http://www.thingiverse.com/> and start printing!

Buy your own filament:

Search for "3D Printer Filament". You can get either ABS/PLA Filament.

Make sure its diameter is 1.75mm!

Change Filament (<http://makergear.wikidot.com/printer-operations>) :

- 1) On Pronterface, set hotend temperature manually to standard temperature (195 for PLA, 240 for ABS)
- 2) Retract several hundred mm until all filament is out.
- 3) Install new filament (Cut its tip at an angle of 30-60°.
- 4) Adjust temperature if necessary (PLA -> ABS : 240; ABS -> PLA: 240; ABS -> ABS: 240; PLA -> PLA: 195)
- 5) Extrude 300mm-600mm of new filament (Until the extruded filament looks clean)