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%](image)

        Infill patterns at 60% – top left to bottom right: Honeycomb, Concentric, Line, Rectilinear, Hilbert Curve, Archimedian 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/](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)