



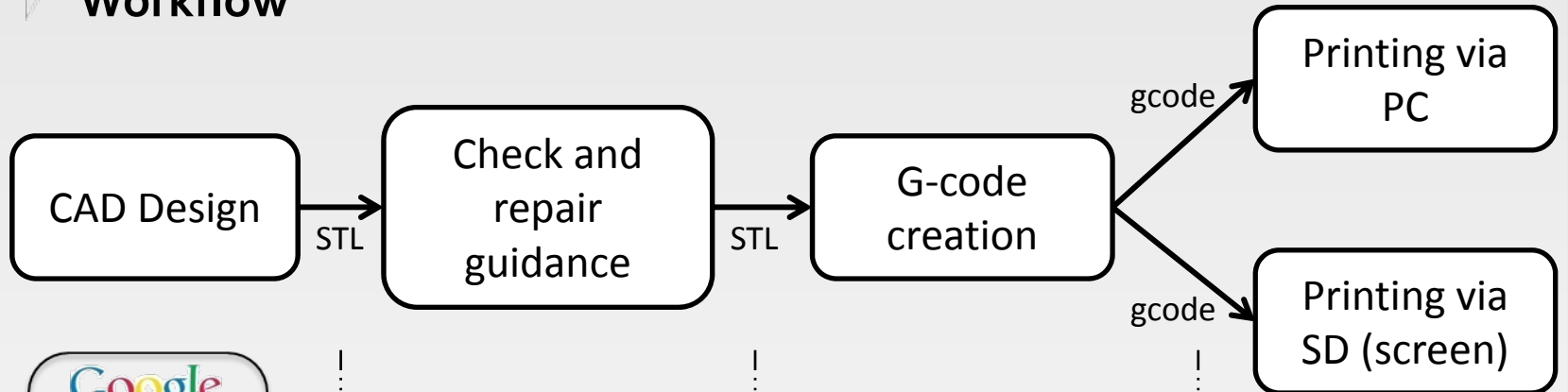
Printing manual

From CAD to your hand!

RepRapBCN Team

- Workflow
- CAD Design
- Netfabb
- Slic3r

Workflow



Open Source



Professionals



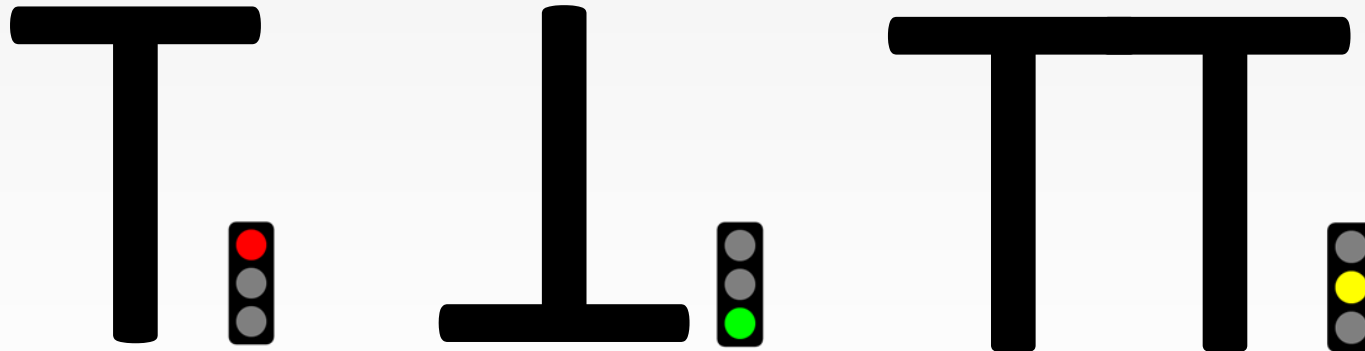
In this manual

▶ **CAD Design:**

- We obtain the designs for the printed pieces from CAD programs.
- Some examples are:
 - Freeware: 123D, GoogleSketchup, OpenSCad...
 - Business Software: Solidworks, Catia, 3DStudio, NX...
- The point is that they export to. STL
- Printing a piece does not depend on the CAD program used but on the STL format.

▶ CAD Design: Prohibited shapes

- In general, all the pieces must finish with flat faces
- The printer does not accept areas that float in the air
- Geometries of **less than 1 mm** are not likely to appear in the printing
- Narrow walls will result in weak parts



▶ **NetFabb**



- Netfabb is a program for the viewing, repairing and transformation of STL files.
- In the present case we will be able to:
 - Place the pieces and prevent them to start printing where they should not.
 - Repair STL that may be damaged
 - Escalar piezas si es necesario

Note: You can download it on their [website](#)

► **NetFabb: platform configuration**

Netfabb let us set the manufacturing volume of our printer.

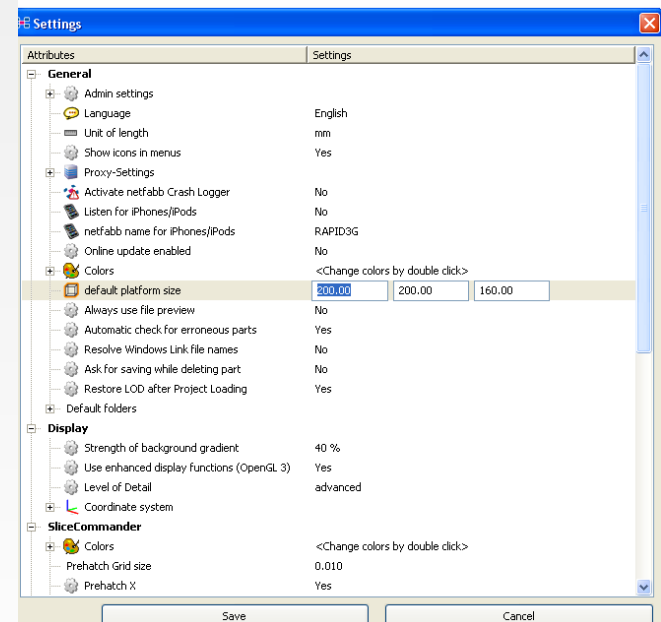
- This will help us to put the pieces correctly.
- It also useful to have an idea about the dimensions of the parts

To configure the platform we must:

- Settings > Settings > Default Platform Size
- Write down 200mm

We have to see the platform:

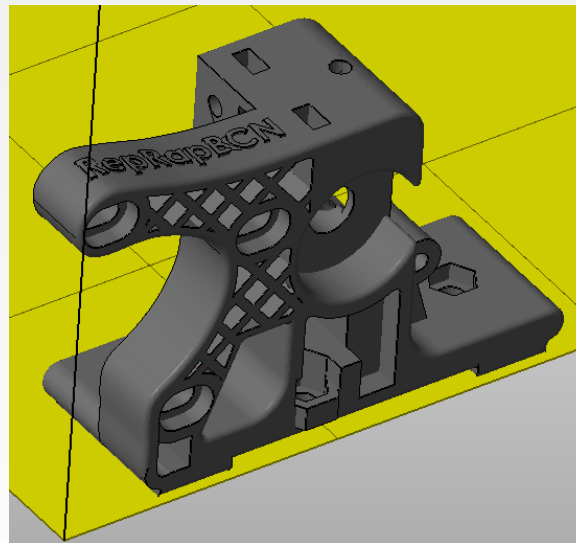
- View > Show Platform



▶ **NetFabb: opening a piece**

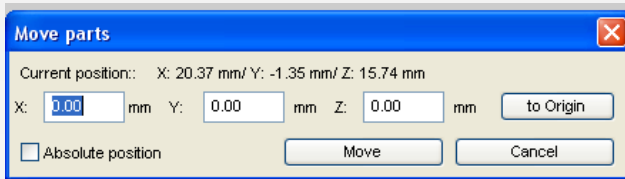


- To open a piece we just have to go to *project > open* or drag a *.stl to the NetFabb window
- The pieces are designed sometimes without following the coordinates the printer needs to start printing. Thus, the piece is disoriented.

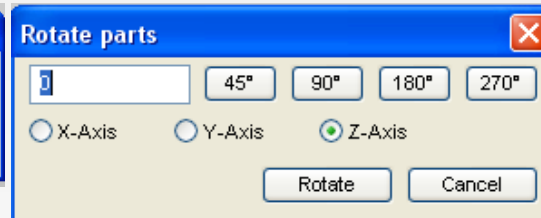


► **NetFabb: parts and position scaling**

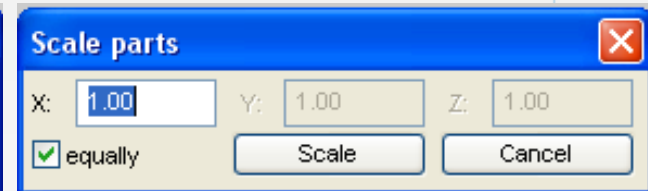
To change the position and scaling of the piece you can use the top menu icons.



It is used to move the pieces according to the coordinate axes. The "to Origin" button sends the piece to the origin, which it will be always made before saving any part



Part rotation according to the axes. Place the piece by placing the face where you want to print on the platform (yellow area)



Scaling according to the axes. To modify a single axis deselect "Equally".

▶ **NetFabb**

- If there's a bad part a warning signal will appear in the bottom left area.



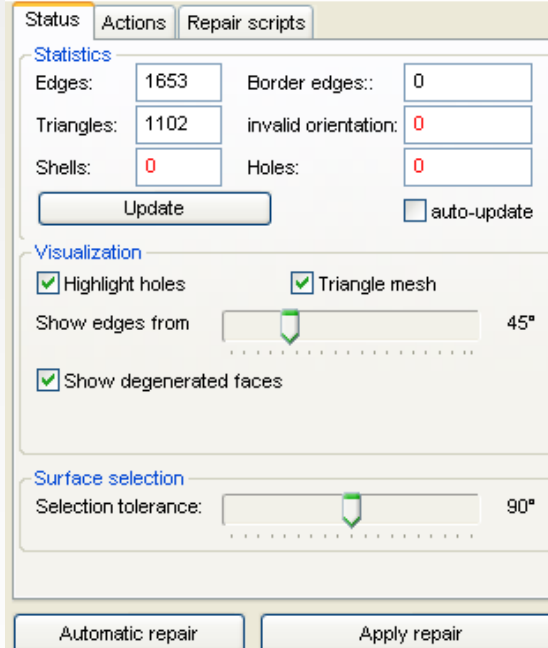
- Netfabb can also repair parts in bad condition using the icon.



- You get in a repair module.

► **NetFabb**

- Damaged faces are red.
- To see all the features from this module watch this [video](#)
- Selecting directly Automatic repair > Default Repair usually solve all problems



The screenshot shows the NetFabb repair interface with the following sections:

- Status** (selected tab)
- Statistics**
 - Edges: 1653
 - Triangles: 1102
 - Shells: 0
 - Border edges: 0
 - invalid orientation: 0
 - Holes: 0
- Visualization**
 - ☒ Highlight holes
 - ☒ Triangle mesh
 - Show edges from: 45°
 - ☒ Show degenerated faces
- Surface selection**
 - Selection tolerance: 90°

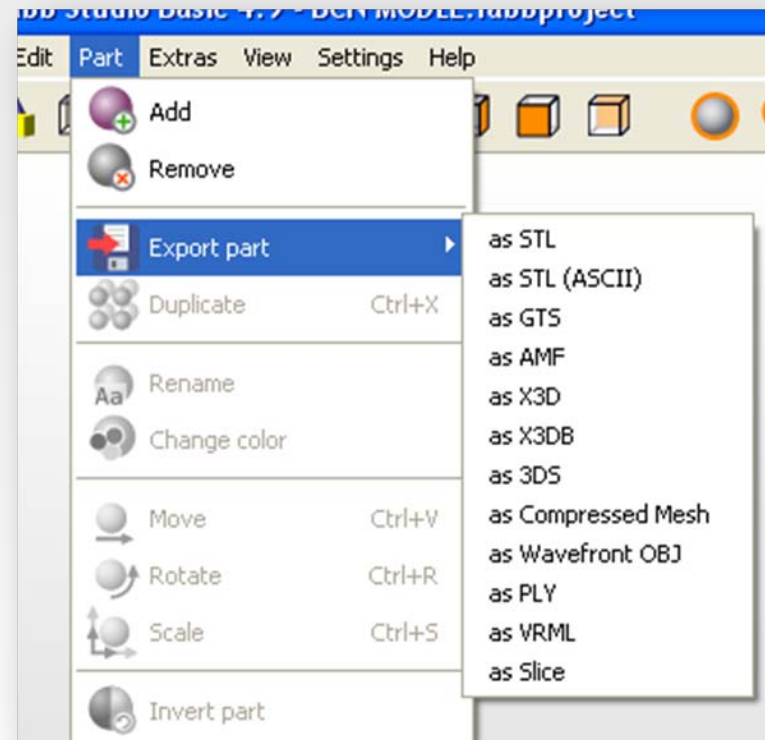
Buttons at the bottom: Automatic repair, Apply repair

► **NetFabb**

Once the piece is repaired and positioned the right way we will export it to a new STL

- No matter what kind of STL we choose.

Now you can open Slic3r





▶ Slic3r

- Slic3r is the program that converts the STL files in the correct format to make the printer start printing (G-code).
- In the present case we will:
- Configure the properties of our printer
 - Set the parameters with which we want to print the pieces
 - With a STL in perfect conditions, **create the file .gcode that we will export to the printer**

Slic3r is not a simple program. We recommend using the settings provided by RepRapBCN and modify them especially just for those pieces that really need it

- To download the latest version of the program you can go to the website of [Slic3r](http://www.slic3r.org).

Slic3r: G-code

What is a G-code?

- It is a text file that contains all the movements and temperatures to run the printer.
- To learn more about these orders click [here](#)
- This file is the one that reads the Arduino; it converts it to impulses and electrical slogans for drivers and resistors in order to print the desired piece.

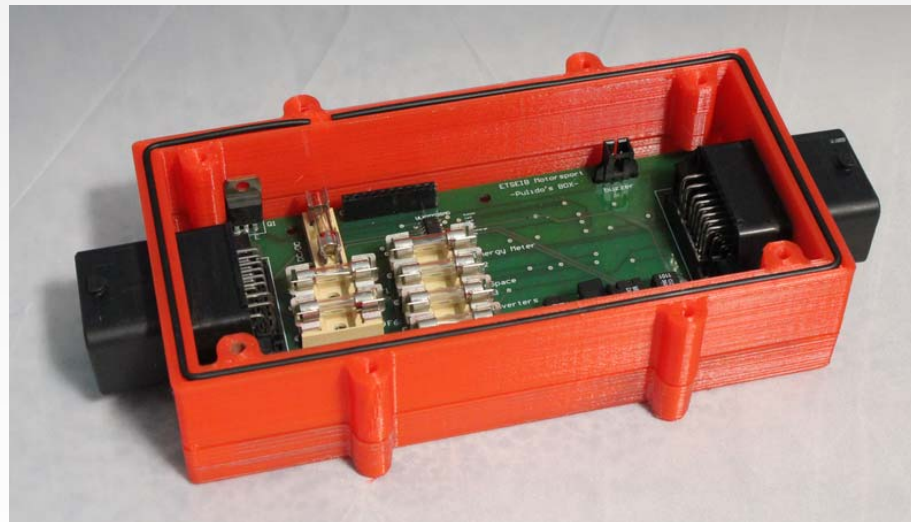
```
Pieza1 - Bloc de notas
Archivo Edición Formato Ver Ayuda
; generated by Slic3r 0.8.4 on 2012-11-24 at 19:49:54

; layer_height = 0.2
; perimeters = 3
; solid_layers = 3
; fill_density = 0.4
; nozzle_diameter = 0.5
; filament_diameter = 2.93
; extrusion_multiplier = 1
; perimeter_speed = 40
; infill_speed = 50
; travel_speed = 150
; scale = 1
; single wall width = 0.53mm

M107
M190 S55 ; wait for bed temperature to be reached
M104 S195 ; set temperature
G28 ; home all axes
M109 S195 ; wait for temperature to be reached
G90 ; use absolute coordinates
G21 ; set units to millimeters
G92 E0 ; reset extrusion distance
M82 ; use absolute distances for extrusion
G1 Z0.300 F9000.000
G1 X62.670 Y90.020
G1 F1200.000 E2.00000
G1 X63.810 Y88.950 F840.000 E2.02410
G1 X64.440 Y88.450 E2.03650
G1 X65.150 Y87.950 E2.04988
G1 X65.840 Y87.530 E2.06233
G1 X67.260 Y86.830 E2.08673
G1 X68.670 Y86.300 E2.10995
G1 X69.520 Y86.060 E2.12357
G1 X71.000 Y85.770 E2.14681
G1 X72.410 Y85.640 E2.16864
G1 X72.810 Y85.630 E2.17480
G1 X107.570 Y85.640 E2.71058
G1 X108.990 Y85.770 E2.73256
G1 X110.470 Y86.060 E2.75580
G1 X111.320 Y86.300 E2.76942
G1 X112.730 Y86.830 E2.79264
G1 X113.460 Y87.170 E2.80505
G1 X114.210 Y87.570 E2.81815
G1 X114.900 Y88.000 E2.83068
G1 X116.130 Y88.910 E2.85426
G1 X117.320 Y90.020 E2.87935
G1 X117.780 Y90.510 E2.88971
G1 X118.300 Y91.130 E2.90218
```

▶ **Slic3r: Settings**

- Unlike the traditional paper printing, 3D printing depends largely on the piece that you want print.
- Each typology or use of the piece will require a custom configuration.
- **Actually there is a setting for each piece.**



▶ Slic3r: Settings

These settings are a starting point.
You can download them [here](#)

- **ST:** Standard configuration with layer height of 0.2 mm for most printings.
- **HQ:** High quality standard configuration with layer height of 0.1 mm for most printings.
- **HS:** Setting designed for fast projects, without losing much quality.
Therefore, a bed height of 0.25 and an infill of 10% is set; only 2 perimeters are made and the speed is increased.
- **SV:** Special setup for container type pieces. There is not infill and the piece is printed spirally with a single surface, improving the surface finish.

▶ Slic3r: Settings

- **WALL:** Special setup for parts very "exotic" geometries, such as those generated by the Voronoi algorithm, with its greatest exponent in the designer Dizingof.
It has no infill and the temperature and speed is lowered to improve the cantilevers.
- **STR:** Special settings for pieces that requires a significant mechanical efforts. The infill is made honeycomb and the layer height of 0.15 mm is decreased to improve the adhesion between the layers



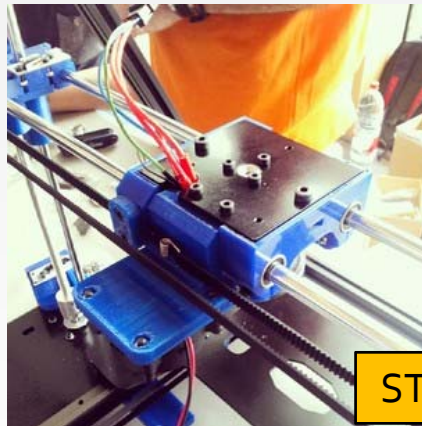
SV



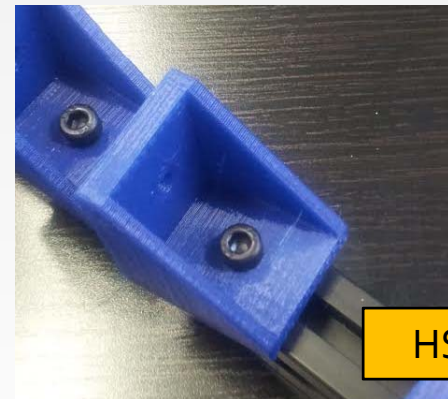
HQ



WALL



ST/STR



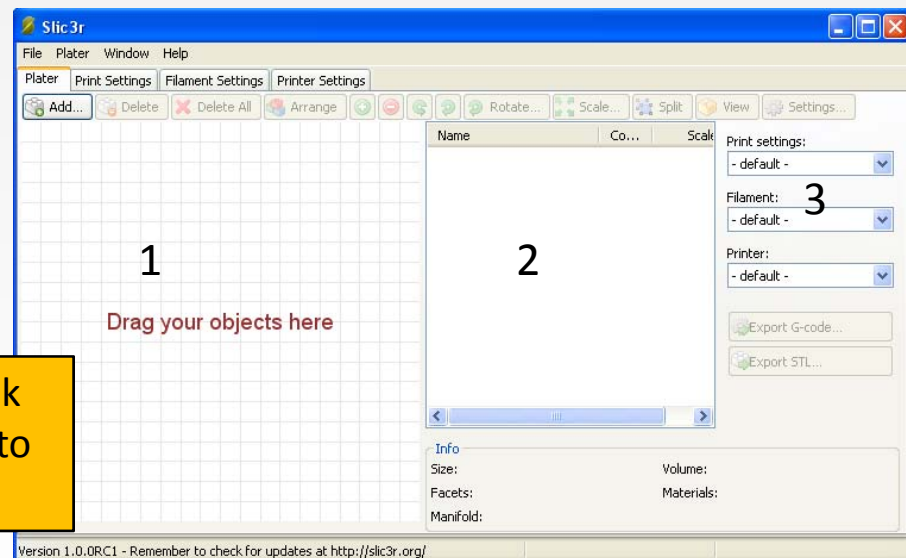
HS

► Slic3r: Previous steps

- Ignore the previous configuration.
 - Initially Slic3r asks a number of printer features that will be used. Omit everything because it will be loaded automatically with the the print settings
- Change display to "expert mode".
 - To do it: File > Preferences > Mode > Expert
 - Close and reopen the program

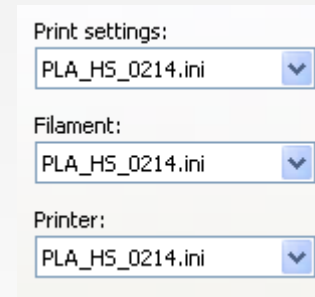
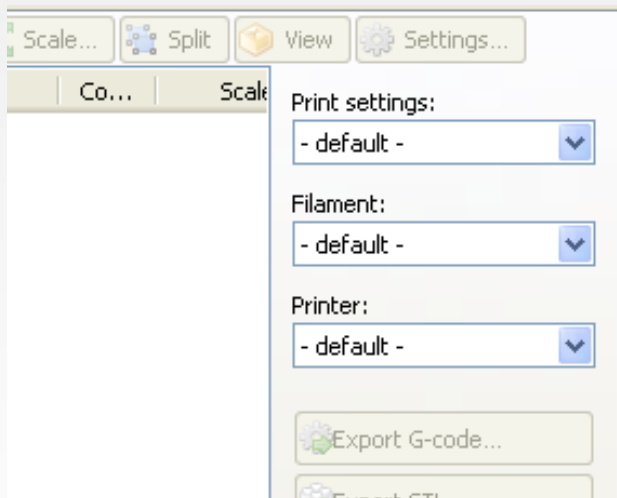
- 1: Platform
- 2: Part list
- 3: Configurations

Slic3r should look like this in order to work with it



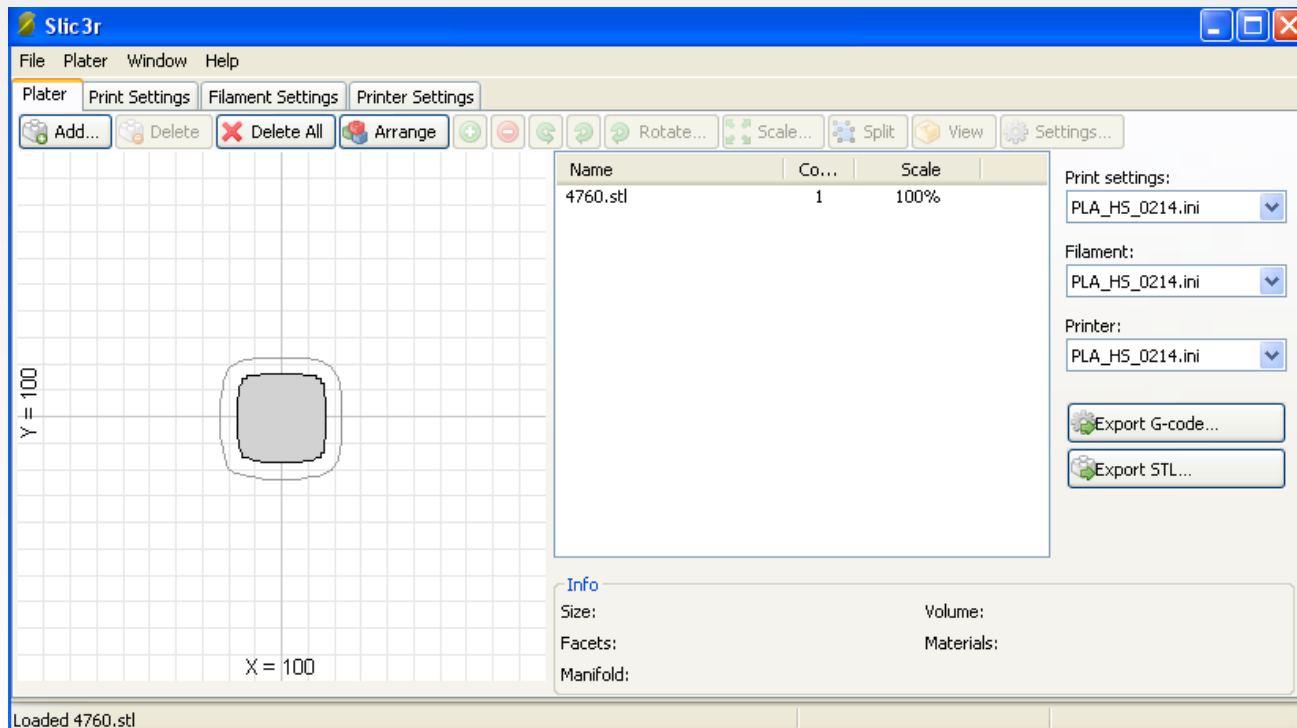
► Slic3r: Printing loading a configuration

- Load configuration:
 - File > Load Config. Select the configuration file (.ino)
 - Slic3r reports that the configurations have been loaded



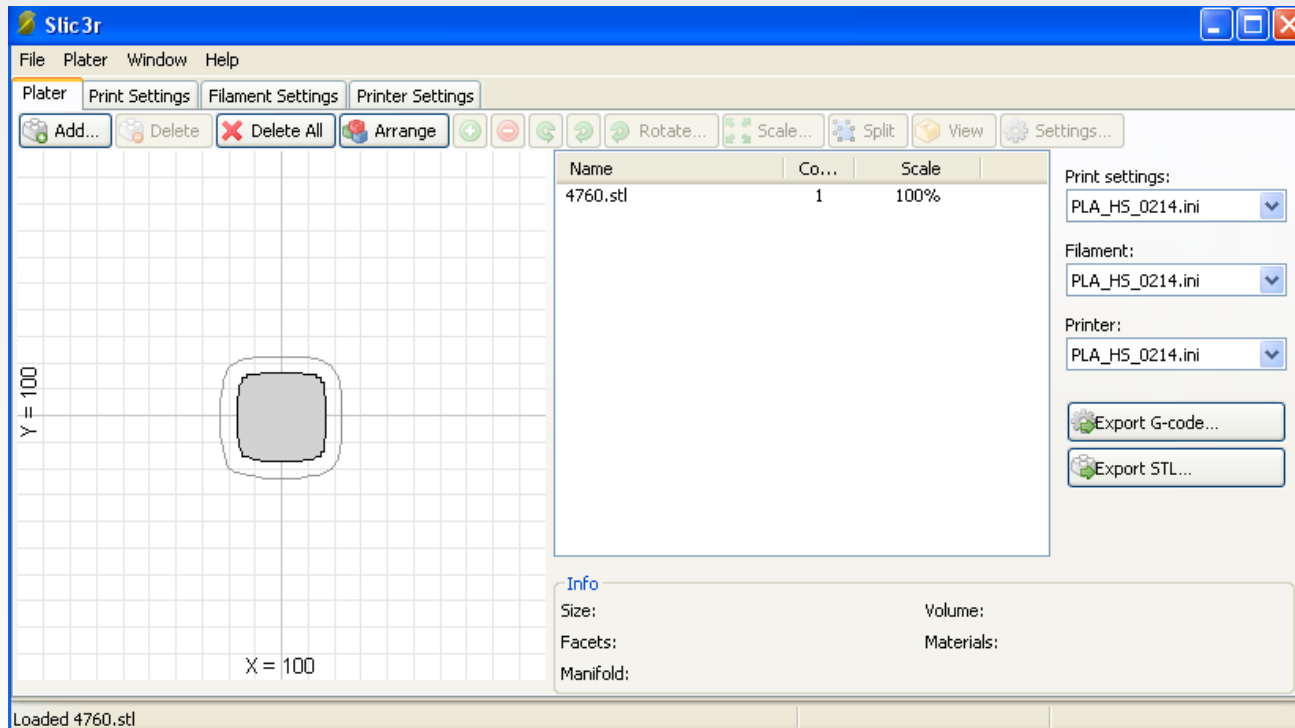
► Slic3r: Printing loading a configuration

- Loading a piece :
 - File > Quick Slice (or Add button). Select the file (.Stl)
- Display piece
 - If we double click on the silhouette a window will pop up



► Slic3r: Printing loading a configuration

- Obtain the G-code:
 - Export the G-code. Select where to save the file.



▶ **Print workflow**

Once the G-code file is available file we have to take it to the printer.

- The file is loaded on an SD card and we have to put the card in the printer reader (on the side of the LCD)
- Printer menu > Print from SD > *file.gcode*

▶ Slic3r: Print settings

Main Printing Parameters: they are the most important parameters that will define the print.

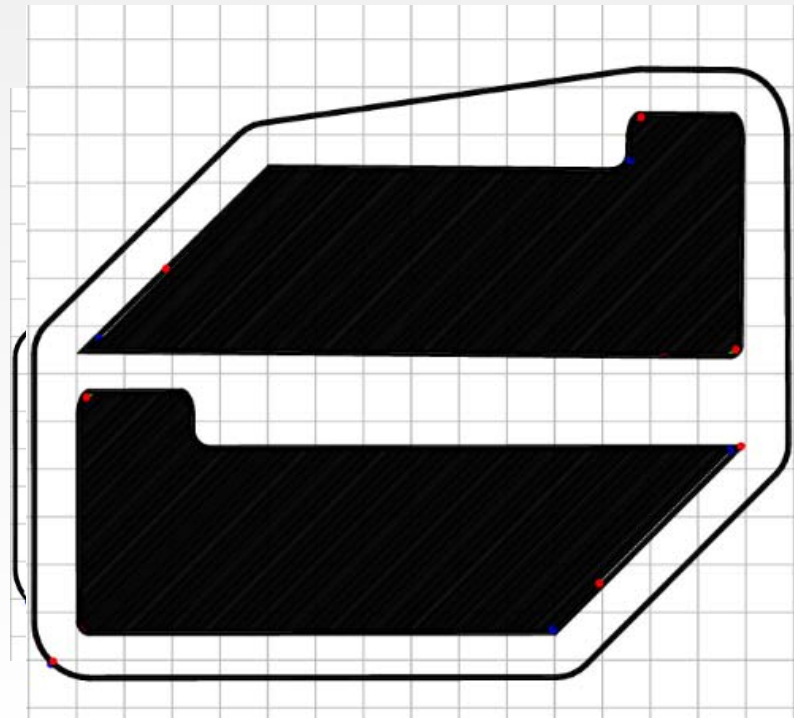
- Layer Height
 - It determines the height of each layer of the piece.
For instance: On a piece of 10 mm high with a 0.3mm layer height would have about 33 layers.
 - Spindle speed
It determines the speed at which the nozzle will move in relation to the printed piece
 - Flow (extrusion speed)
Amount of material that is pushed by the nozzle per unit time

The parameters are dependent upon each other.

For instance: *A higher speed of the spindle to a same height layer will cause (or it should cause) an increase in flow.*

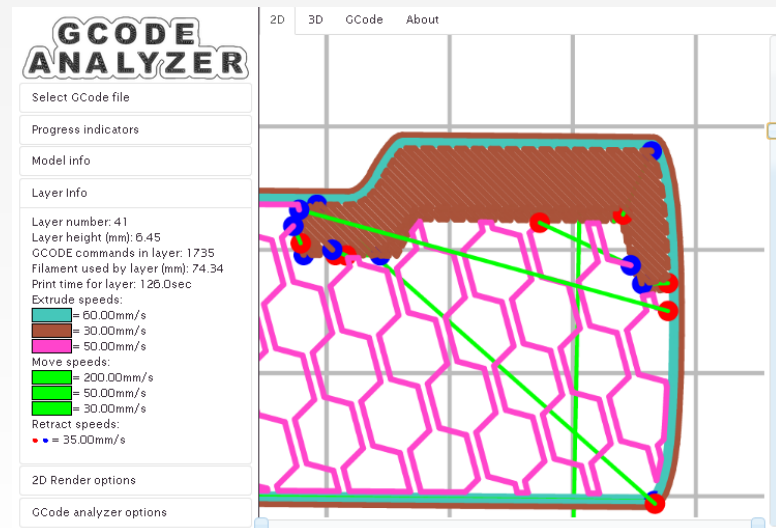
► **Slic3r: Construction strategy**

- Autohome
- Skirt: Purge the extruder before the piece
- Print layer: :
 - Perimeters: Outside of the piece (the skin of the piece)
 - Infill: Inside of the piece
- Piece delivery



► Slic3r: construction strategy

- Gcode analyzer (gcode.ws) is a G-code file analyzer. You will be able to:
 - Know the estimated printing time
 - Know the estimated time for each layer
 - Visualize the trajectories for a layer (bottom bar)
 - Visualize the different layers (right sidebar)
- Works online
- It is an indispensable tool if you go into detail on 3D printing



▶ Slic3r: tabs



- On the second tab "*Print settings*" print settings are managed
- In "*Filament settings*" material properties and working temperature is adjusted.
- Finally, in "*Printer settings*" some printer parameters are managed.
- A detailed guide to all the Slic3r parameters can be found on the bottom of [this page](#)
 - (We are developing a more lightweight guide...)



We hope you have enjoyed the experience!
Do not forget to follow us on Twitter, Facebook and
Google+

RepRapBCN Team