RepRap Workflow

When working with the RepRap (and similar 3D printers) the workflow is the following:

CAD/Model → Netfabb → Slic3r → PrintRun (Pronterface)

Brief Overview

CAD/Model

This is the beginning of the 3D printing process. Here you have either generated a model via various Computer aided design (CAD) programs or acquired a 3D model via other resources (e.g. Thingiverse, GrabCAD, etc.). One thing to remember in working with the RepRap 3D printers and the various other derivatives is that the final format for the object you would like to print needs to be an STL.

downloads:
- Open SCAD (CAD for programmers, free and open-source)
- 123D Design (3D modeling for the DIY community, easy to learn, but still useful)
- Rhino (very powerful but user-friendly 3D modeling, free demo)
- ReconstructMe (free 3D scanning with Kinect, Windows only, also requires Kinect SDK, more info)

Netfabb

Netfabb is a free program that allows us to do a preliminary check on the "3D printability" of our model. In the event that our model is problematic, Netfabb has the tools to fix most of the issues. It's always a good idea to run your model through before getting in front of a 3D printer.

downloads:
- Netfabb Studio Basic (free STL repair)

tutorials
- Netfabb - Basic Repairs
- Netfabb - Basic Trasnformations

Slic3r

The 3D printing process is a series of stacked 3D layers, which ultimately means stacks of 2D drawings/toolpaths. To convert your 3D model into these toolpaths that the 3D printer can then execute (also referred to as G-Code), we need to use a slicing program. For our purposes, we will be using Slic3r as our tool path generator. We will not be getting into the fine details of Slic3r, there will be "known-to-work" defaults saved on the computer and available for you to download.

downloads:
- Slic3r
- Slic3r configuration files for MICA dFab printers (zip)

PrintRun (Pronterface)

This is the last stop before the 3D printer springs into action and creates your object. This is how we will interface/control the 3D printer prior to (and on some occasions, during) printing. In Pronterface, we can set temperatures, jog the axes, load G-code, preview G-code and initiate print jobs.

downloads:
- Pronterface

In-depth Slic3r and PrintRun

When you open Slic3r, your window should look like this.

You should have the drop down menus located at the bottom of the window, if you do not, go to "File" → "Preferences" → Select "Exper" from the drop down that currently reads "Simple." This allows for all the settings in the configurations to be applied to whatever it is that you are slicing:
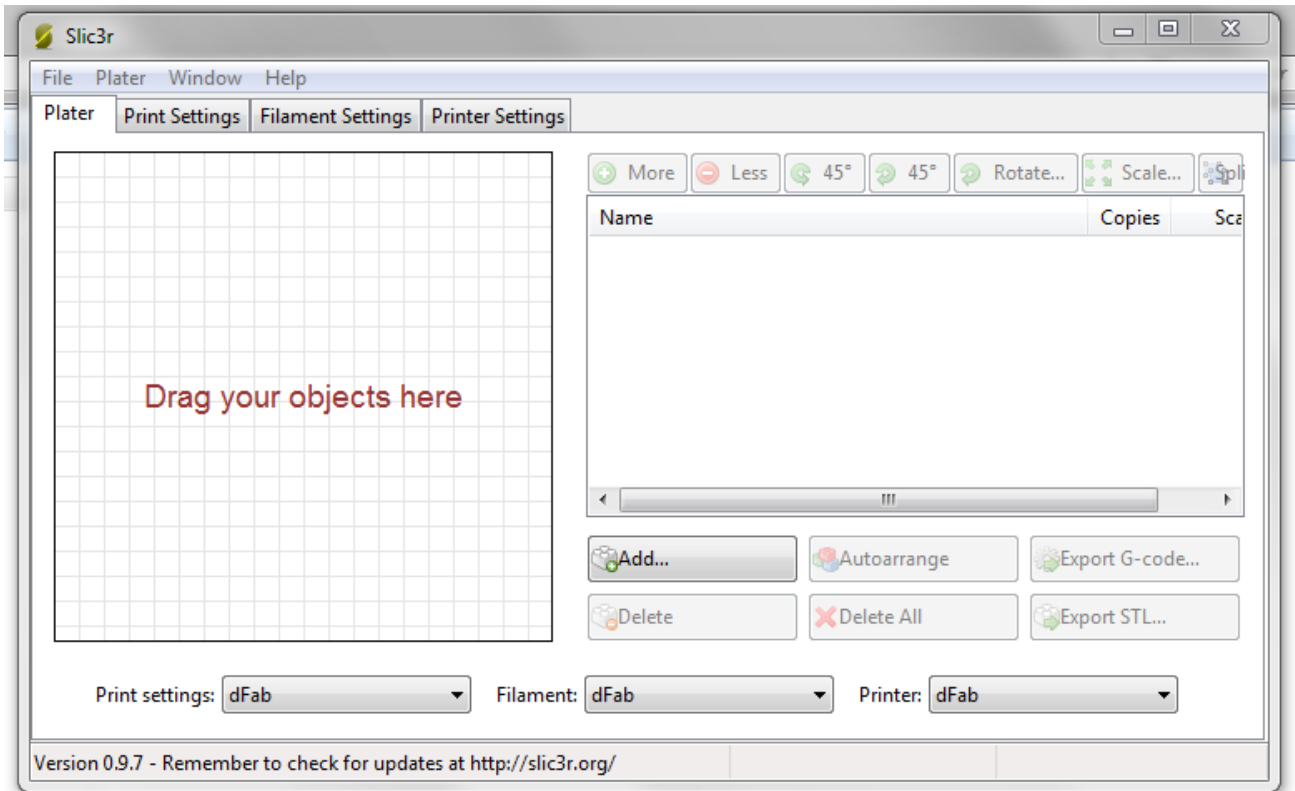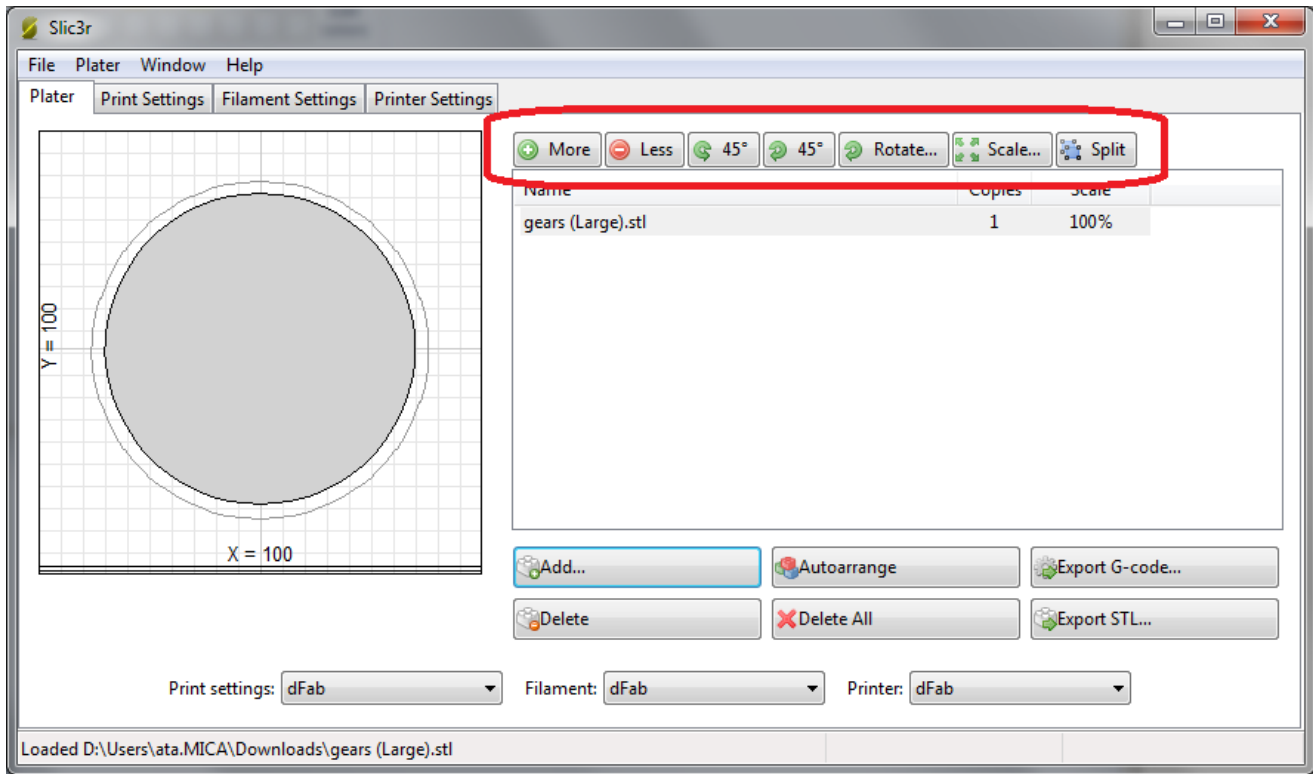

First load the configuration file for the printer you are planning to use.
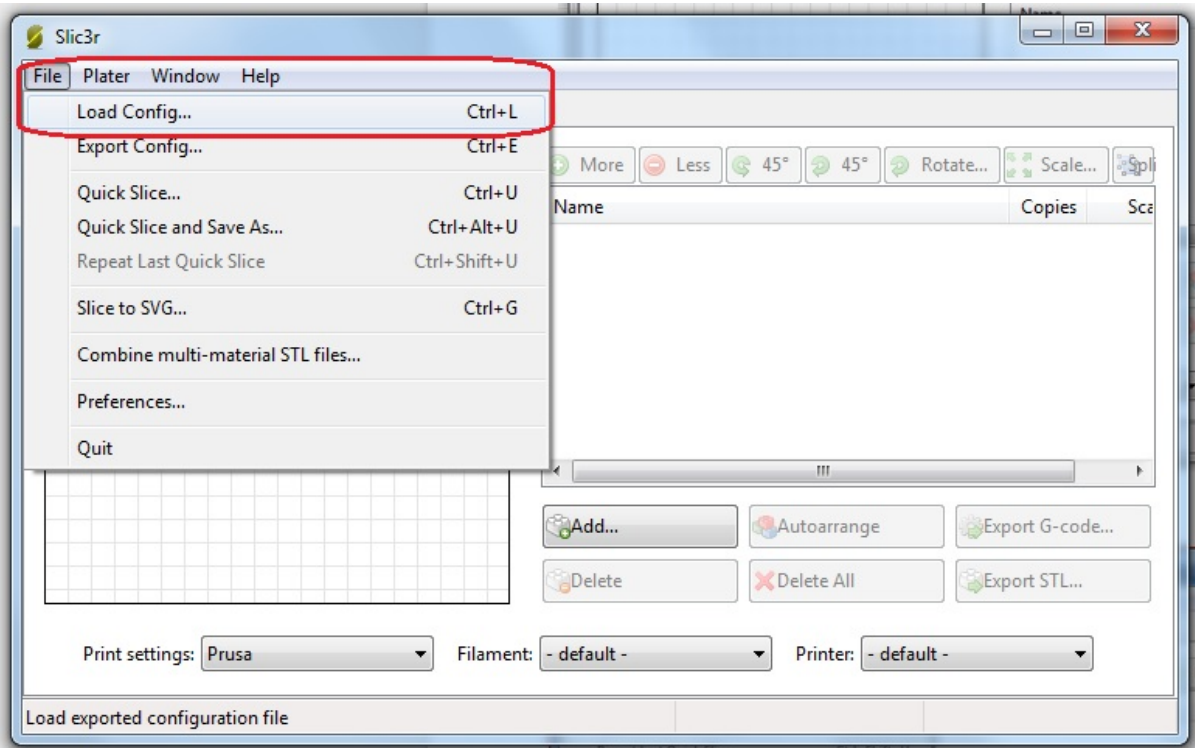
To add your STL to Slic3r, press the "Add" button.
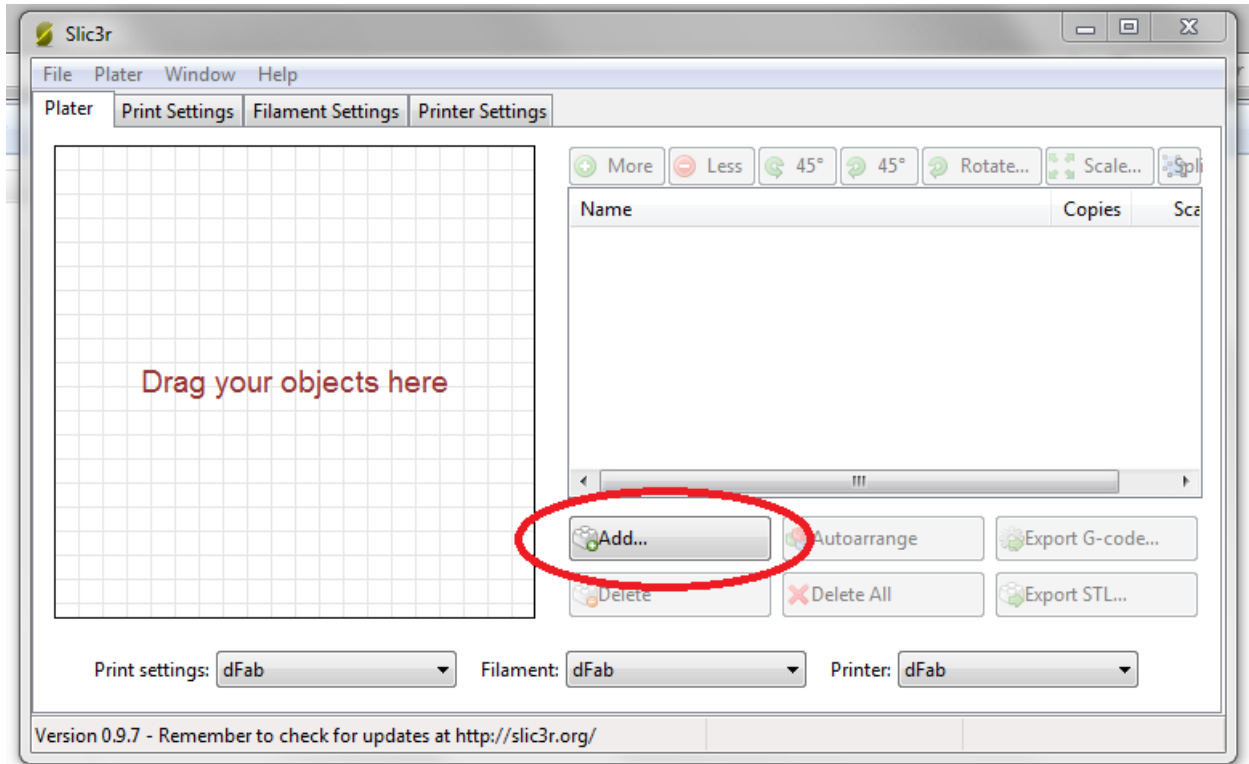
Now that your object is in Slic3r you can do basic manipulations such as add copies, rotate 45 degrees (or an angle of your choosing), scale objects or split objects.

After you have made the manipulations you wanted or you are ready to generate the G-Code, press the "Export G-code" button nears the bottom, right corner.
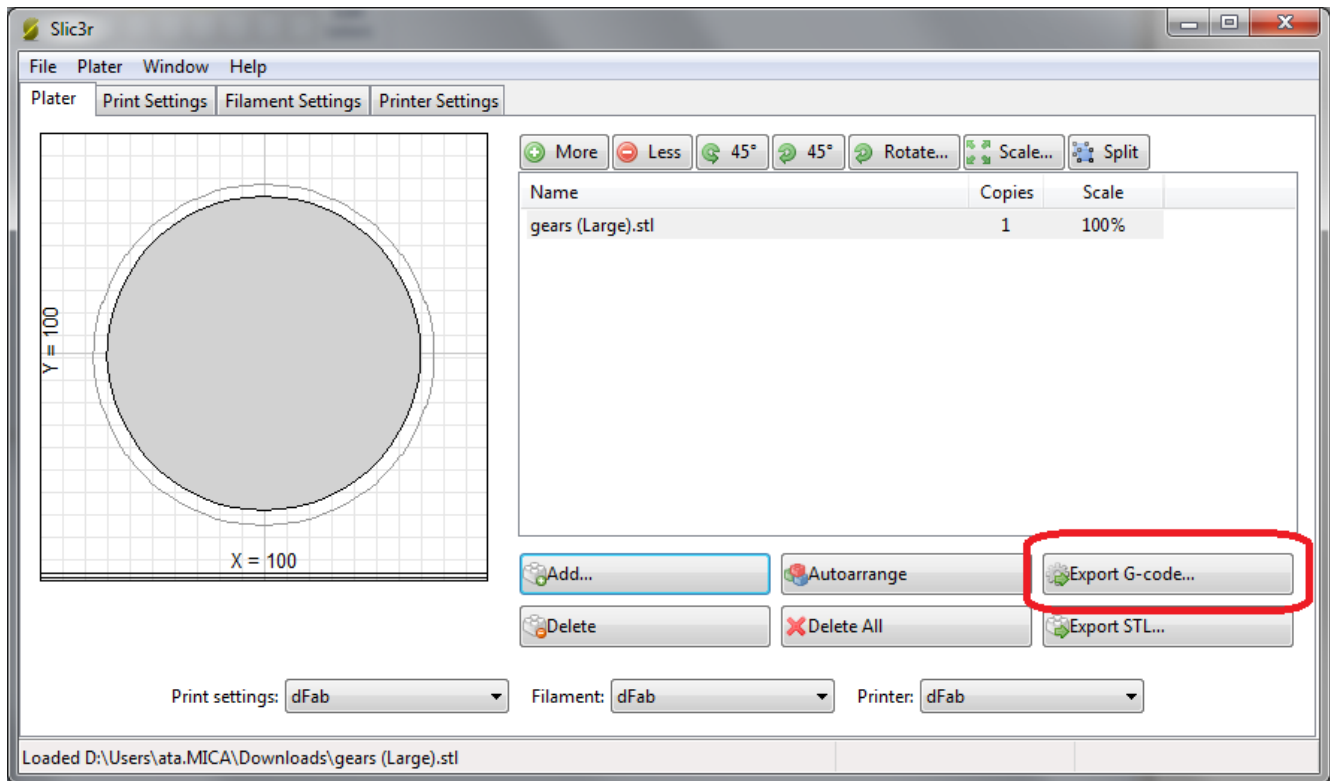

You will be prompted to specify the location in which you want the G-code to be exported, by default it saves to the same location you STL is currently in. Save there or choose a new location. Slic3r will now begin to create the toolpaths according to the settings for the "dFab" profile.
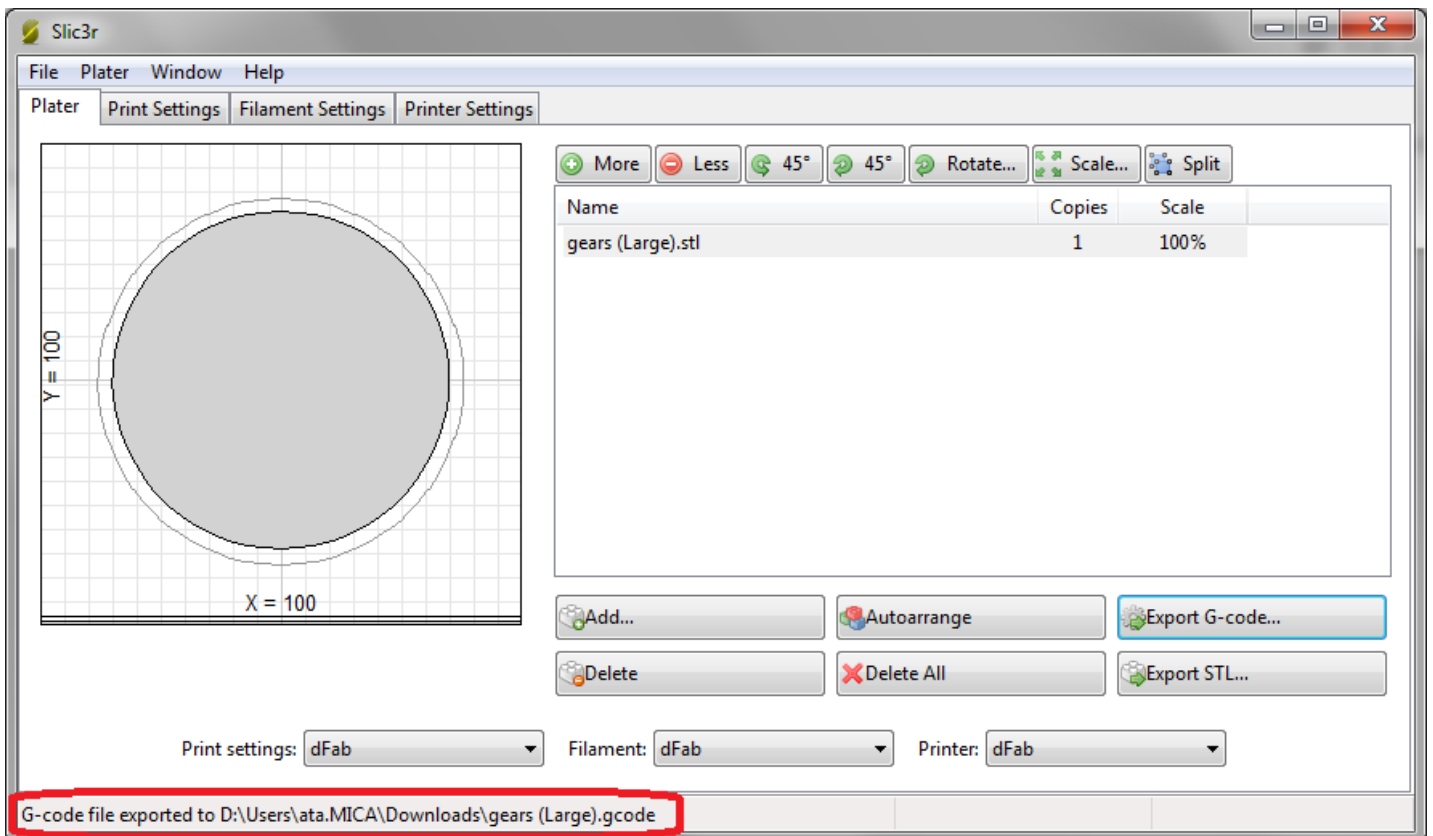
When Slic3r is complete with slicing, it will give you the location of the G-code that was generated. We are now ready to move onto Pronterface.

When you first open Pronterface, you will be greeted with this window:
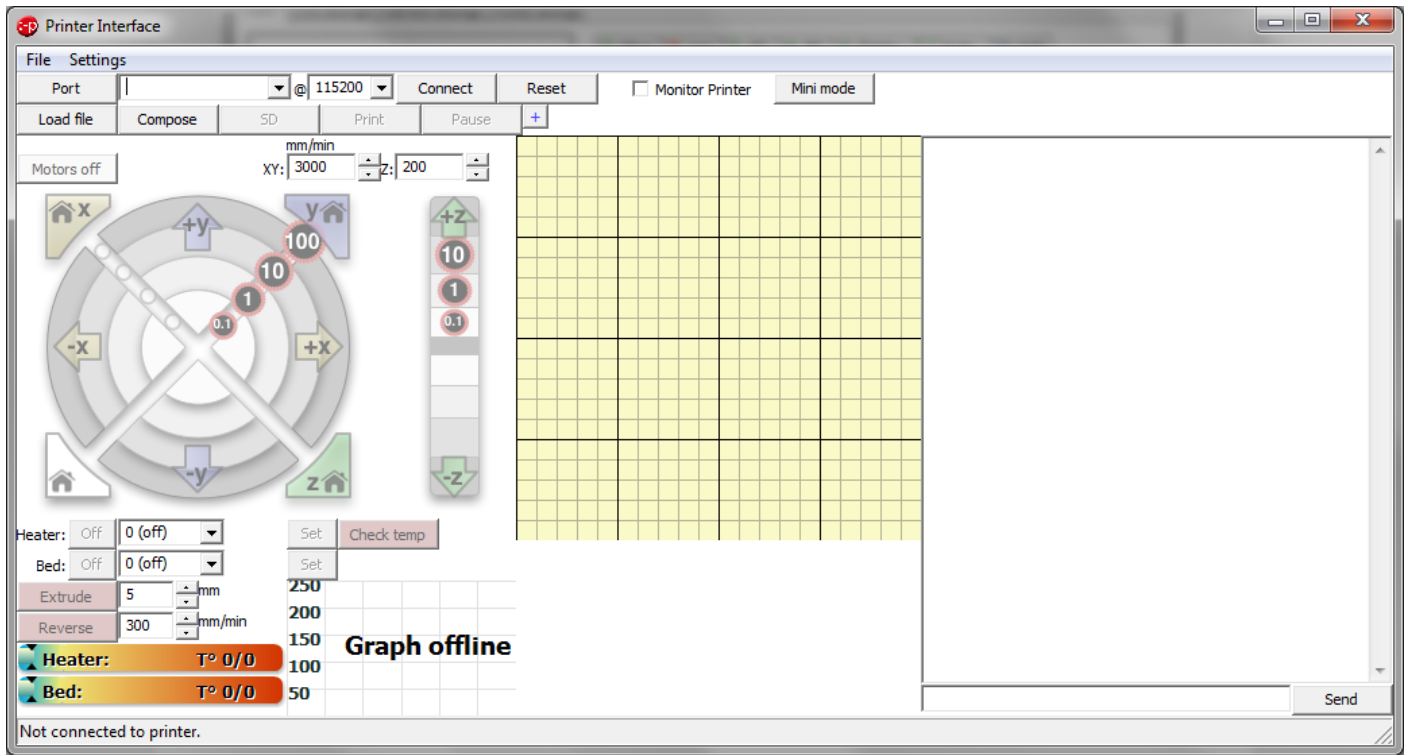
Let's go over the interface by section, first is the "Communications" section (highlighted below). Here is where we tell Pronterface what port our 3D printer is connected to and at what speed communications between the computer and it should be (for the most part it should be set with the right values every time).
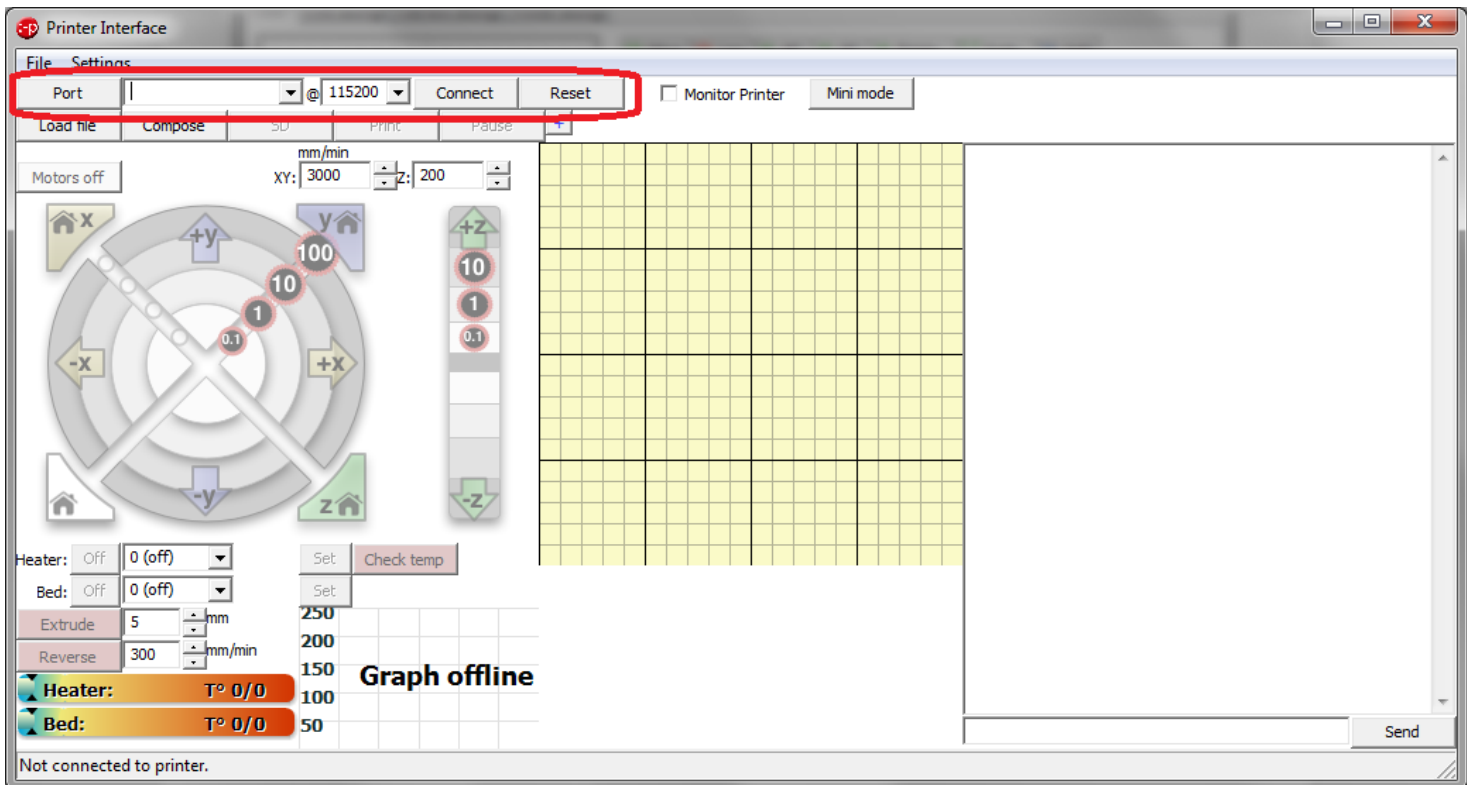

Below the "Communications" area is the "File" tools. Here you can load files to be printed (from the computer of from

SD card option), initiate a print or pause a print.
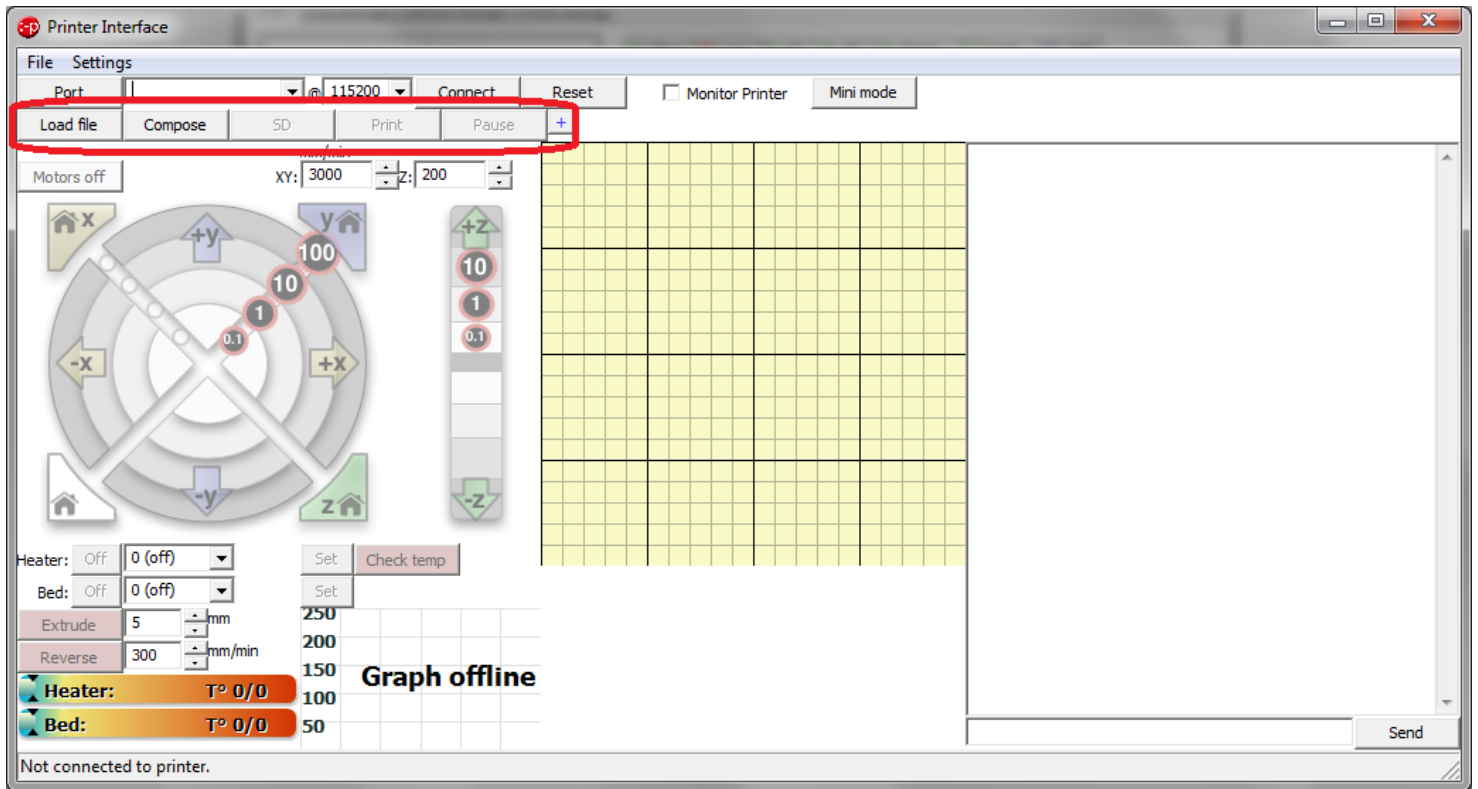
Below the "File" section is the "Operational" section. Here you can jog each axis or home each/all axes. You can see the increments that you can move, from 0.1mm's to 100mm's, in either the positive or negative directions. Speeds for each move can also be set here, as well as disabling the motors when they are not in use (handy when you want to manually move things!).
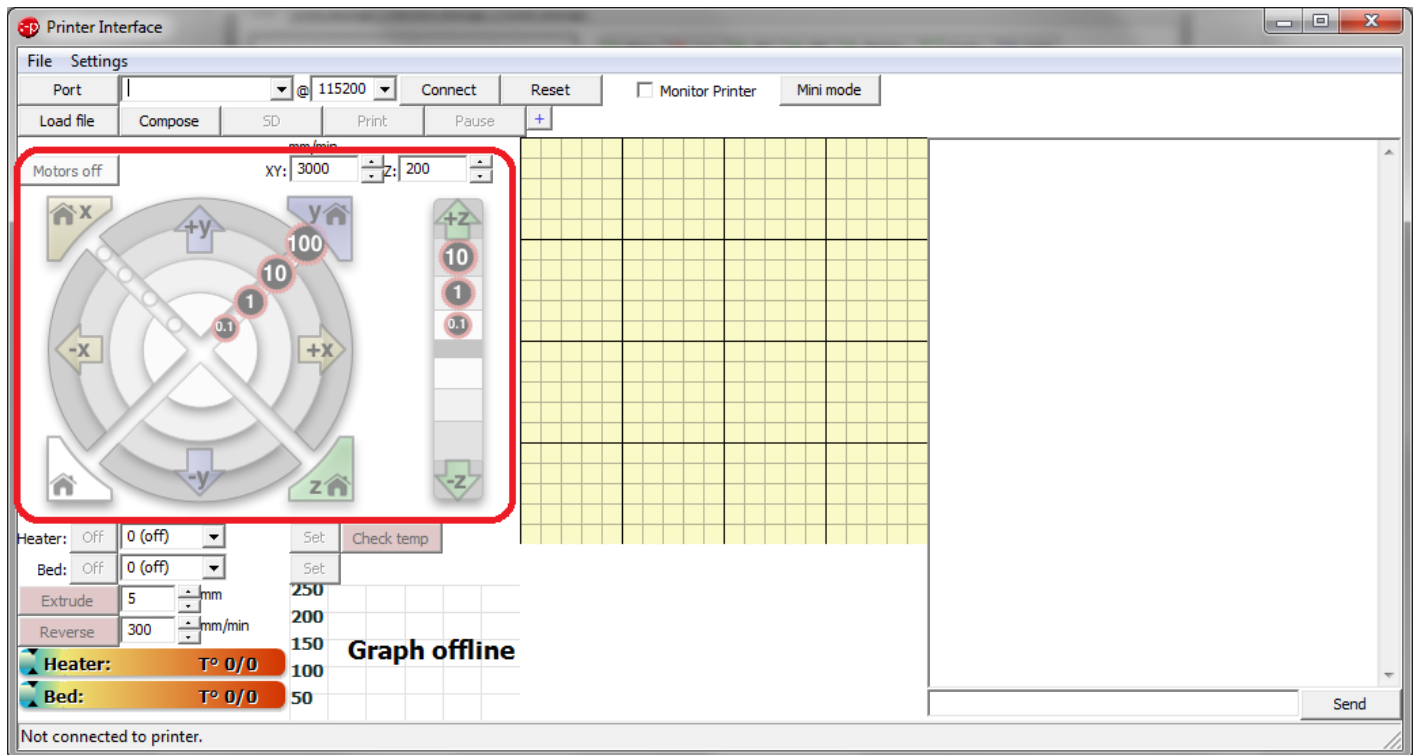

Next thing below the "Operational" section are "Temperature/Extruder" tools. Here you will be able to set the temperatures for the Extruder and the heated bed, move the extruder and get temperature information. You will use this

to pre-heat the heated bed as well as set the temperatures of the printer prior to sending a print job. This is also where you will prime the printer before a print job to ensure optimal flow rate. It is also where you will get quick, easy to digest data on the temperatures of the extruder and heated bed (whether it's up to temp and stable or not being the major "thing" of interest).
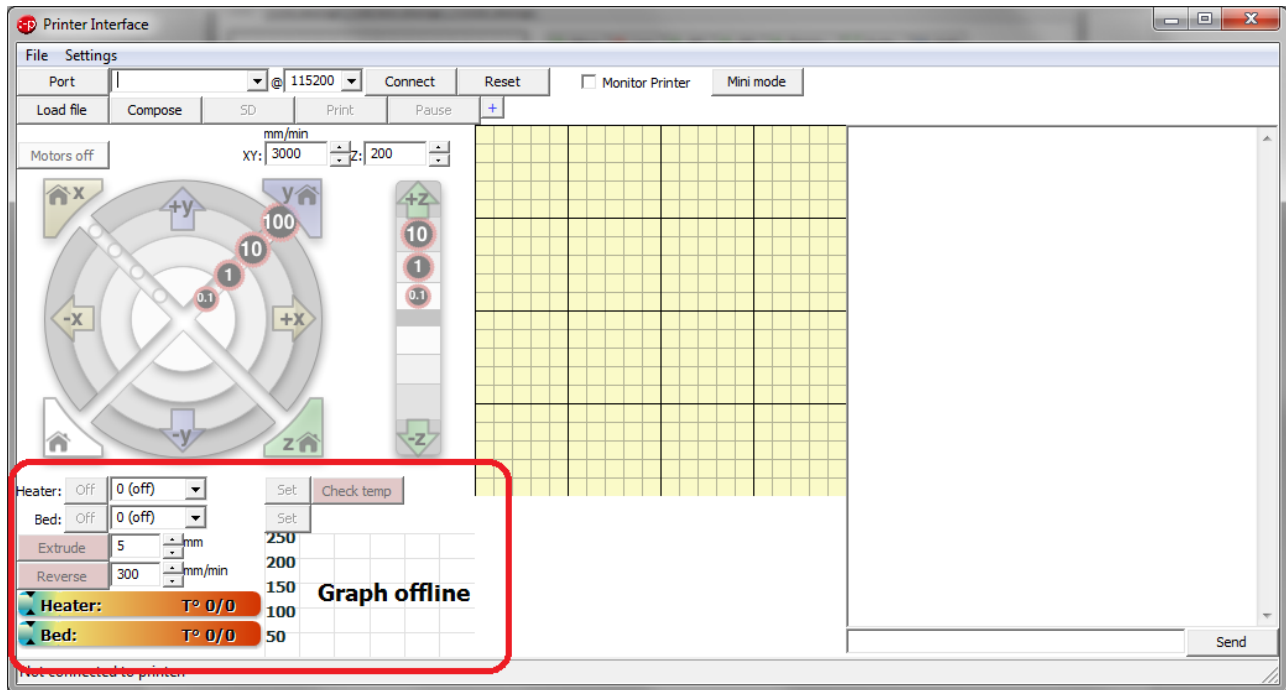

Next we have the "G-code preview" window. Once a G-code file is loaded you will be able to click on the window and scroll through each layer of the model. This is a really good habit to form to ensure your prints turn out in an expected manner.
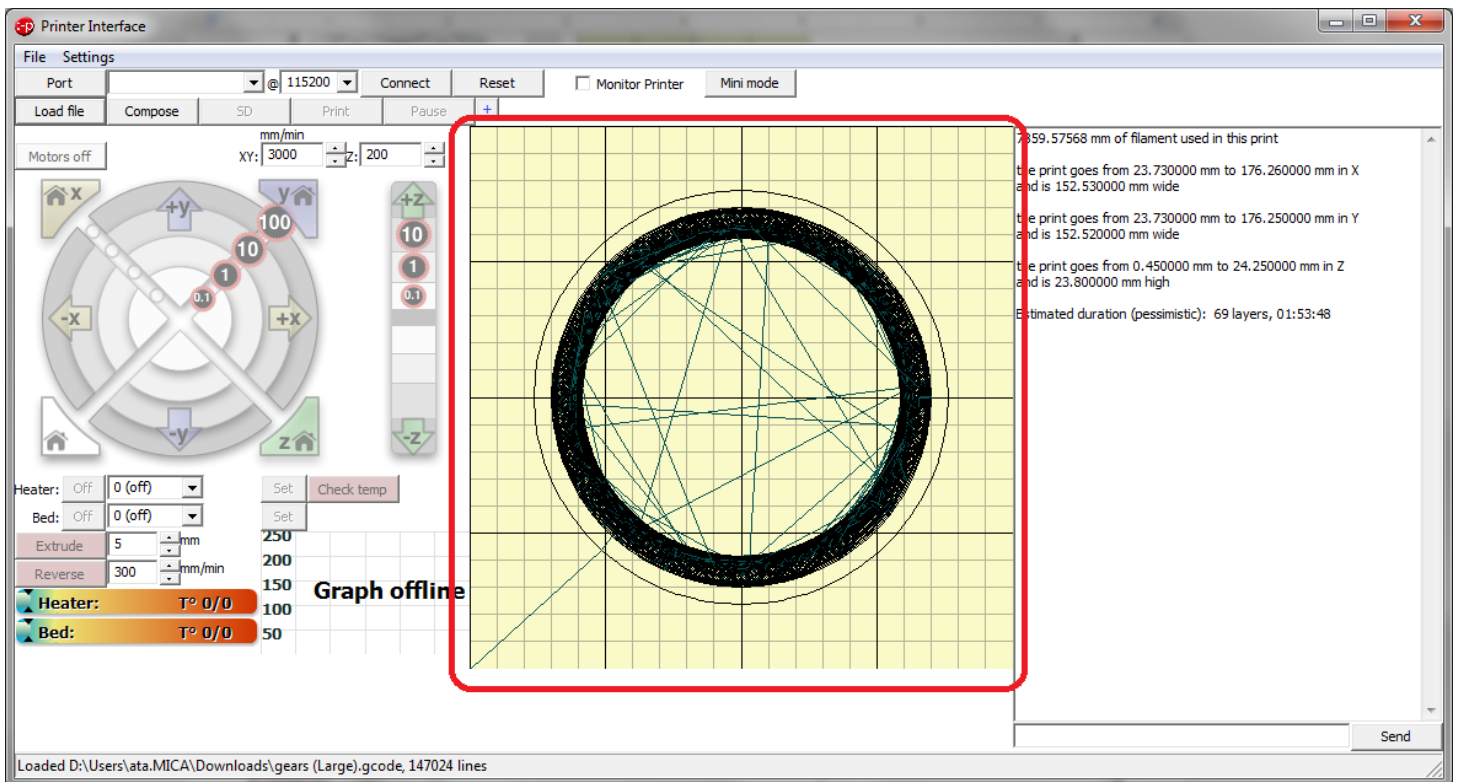

And the last area of interest is the "Console" window. Here is where you will see information such as success of connection, command acknowledgements, print time estimates and so on. It is pretty much your window into what the

3D printer is interpreting every time you press a button. (You can also send G-code commands here too)
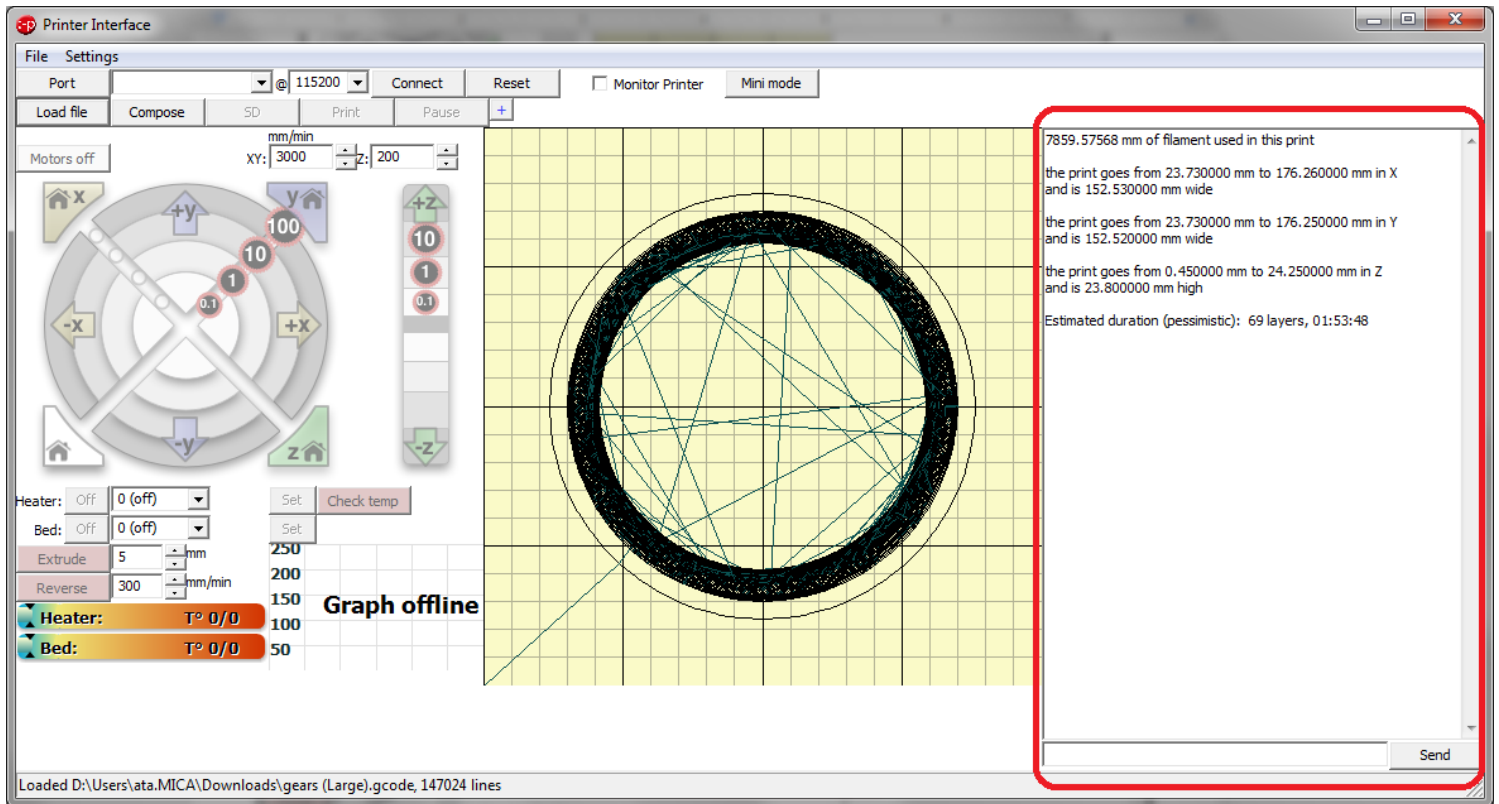
There are a couple features/buttons we did not touch on, those being the "Monitor Printer" check box and the "Mini mode" button. Checking "Monitor Printer" will have the computer stream every tidbit of information the printer is seeing (mainly it will constantly spit out the temperatures and the PWM of the heater on the extruder). The "Mini mode" button will shrink PronterFace down to the following:
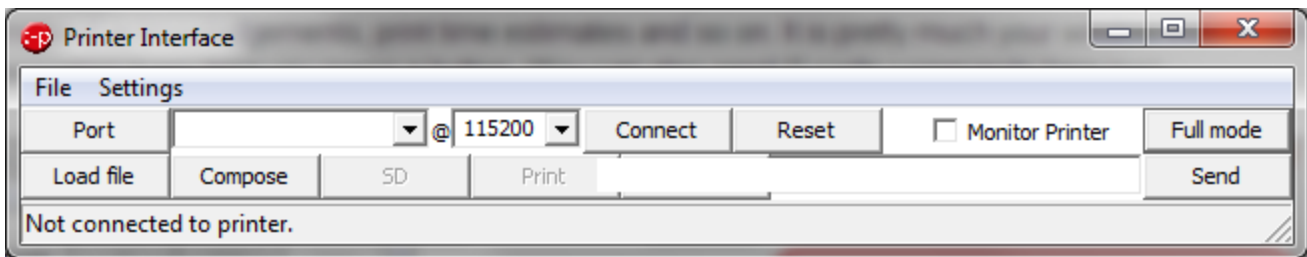
It'll just give you access to the bare essentials, no G-code preview, "Console" window or any of the other graphical features.

And that is it for Pronterface! So, in summary the abbreviated work flow breaks down as follows:
1. Generate 3D model via CAD or 3D model repository
2. Open the model in Netfabb to ensure its 3D printability
3. Use Netfabb to fix any issues
4. Take model and slice it via Slic3r
5. Load model into Pronterface
6. Preview toolpaths
7. Use Pronterface to prep the printer for print job (set temps, home axes, etc.)
8. Print!

And that's it in a nutshell! Congrats and happy printing!