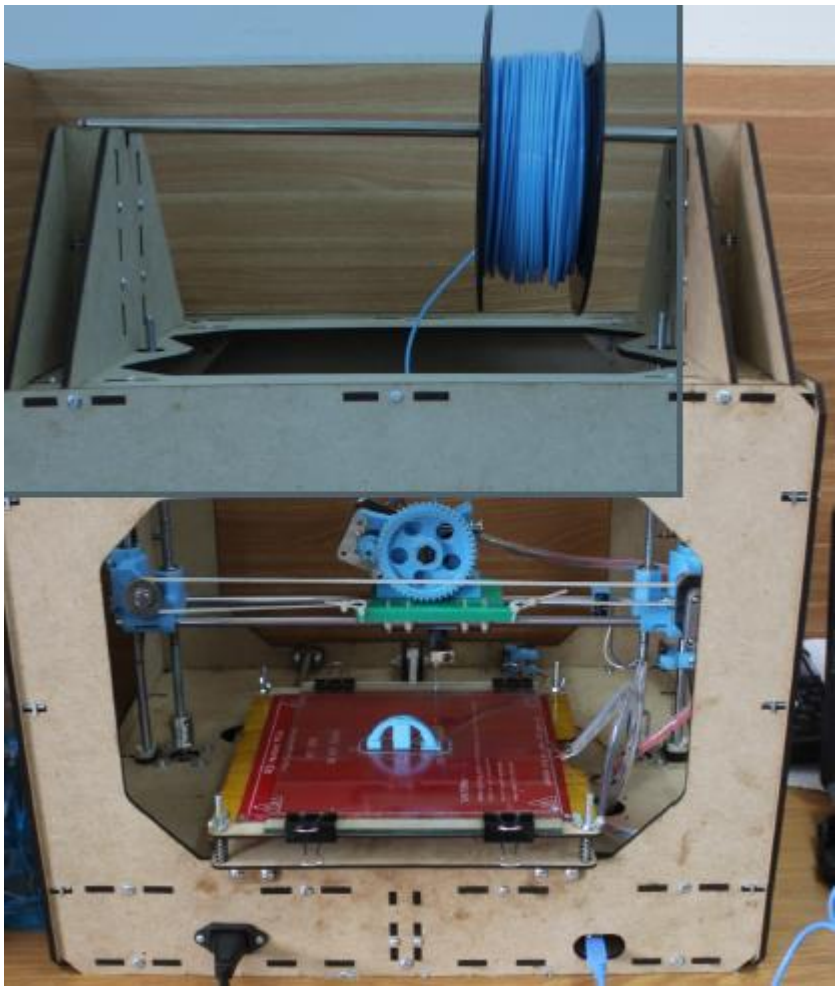
Digital Object Maker

FDM 3D Printer

User Manual



Introduction

This User manual is designed to start using Digital Object Maker 3D Printer in the right direction. Even if you are familiar with other 3D printers, it is essential that you read through this manual.

Specifications

Printing

Print Technology: Fused Deposition Modeling
Print Volume: 20 x 20 x 20 cm.
Layer Resolution: 100-250 micron
Filament Diameter: 3mm
Nozzle Diameter: 0.45mm

Software

Interface software: Pronterface
Slicing software: Slic3r
File Types: STL

Electrical

AC Input: 220 V, 50 Hz
Power Requirements: 12 V DC and 30 Amps
Connectivity: USB

Mechanical

Chassis and body: MDF wood
XYZ Bearings: LME8UU Linear bearing
Stepper Motor: 1.8° step angle
1/16 micro stepping

General

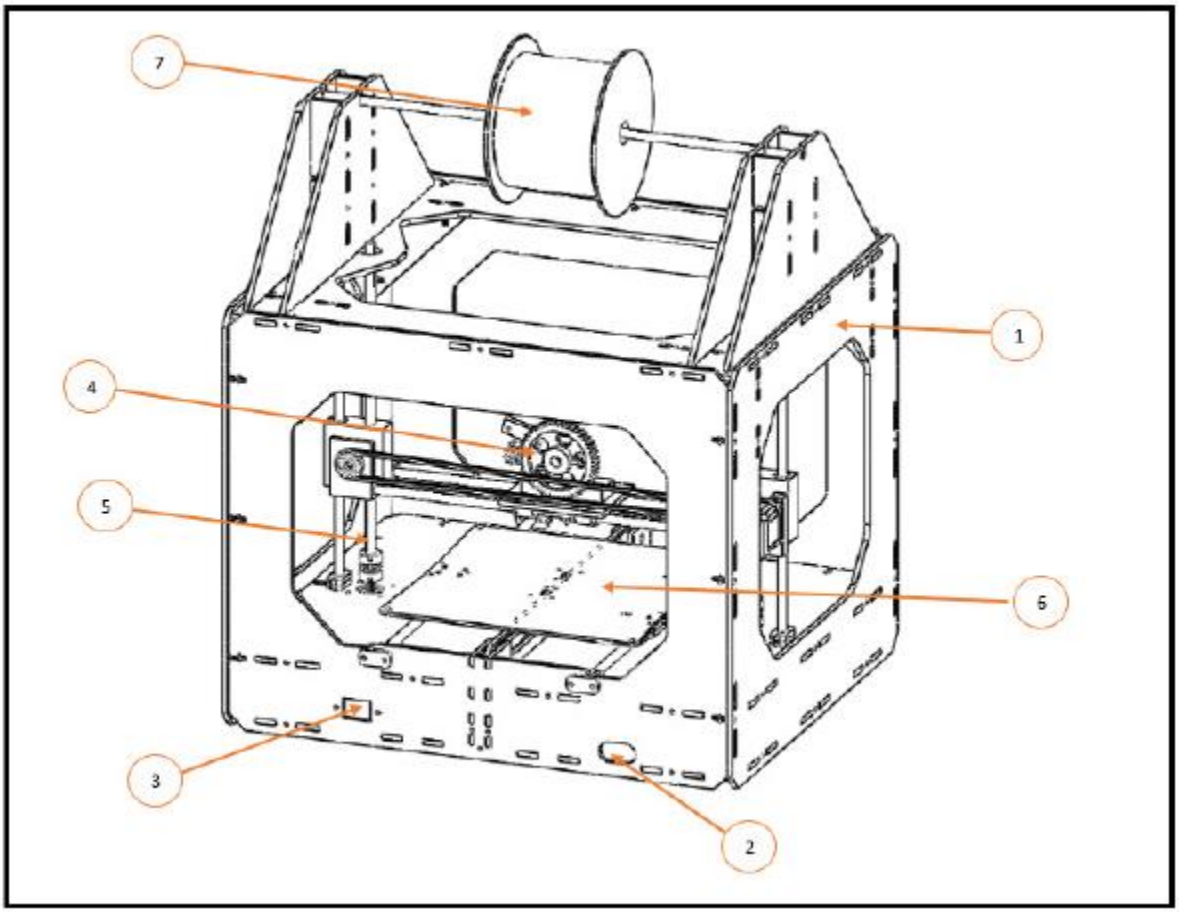
Frame Dimensions: 49x42x55 cm^3
Printing Material: PLA
Filament Diameter: 3mm

How it is work

The DOM makes solid, three-dimensional objects out of melted PLA Filament. Your 3D design files are translated into instructions for the DOM and sent to the machine via USB cable.
Then the DOM heats the PLA Filament and squeezes it out through a nozzle to make a solid object layer by layer. This method is called Fused Deposition Modeling [FDM].

Printer components

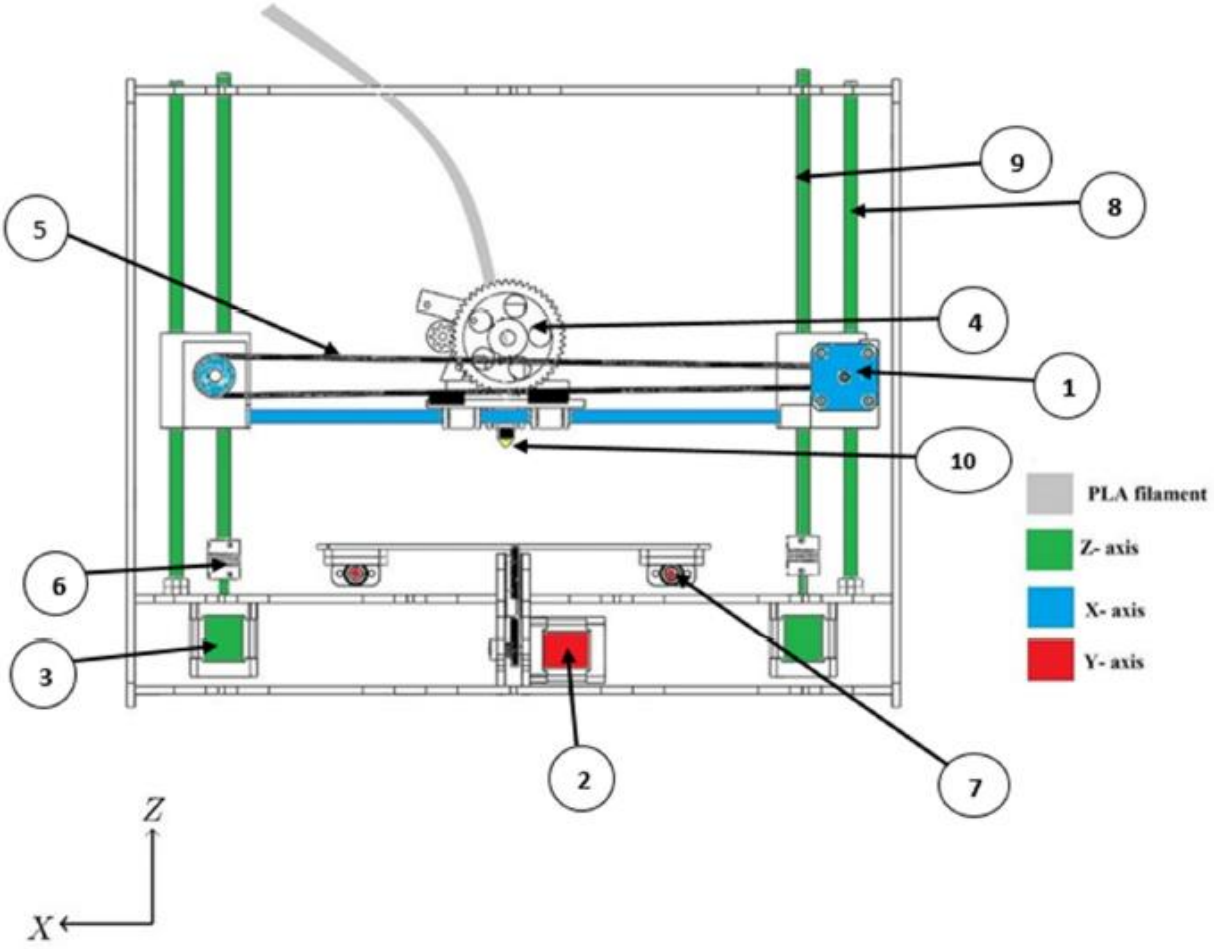
Main diagram



1. Frame
2. USB cable inlet
3. Power cable inlet
4. Extruder
5. Z axis threaded rod
6. Print bed
7. PLA filament

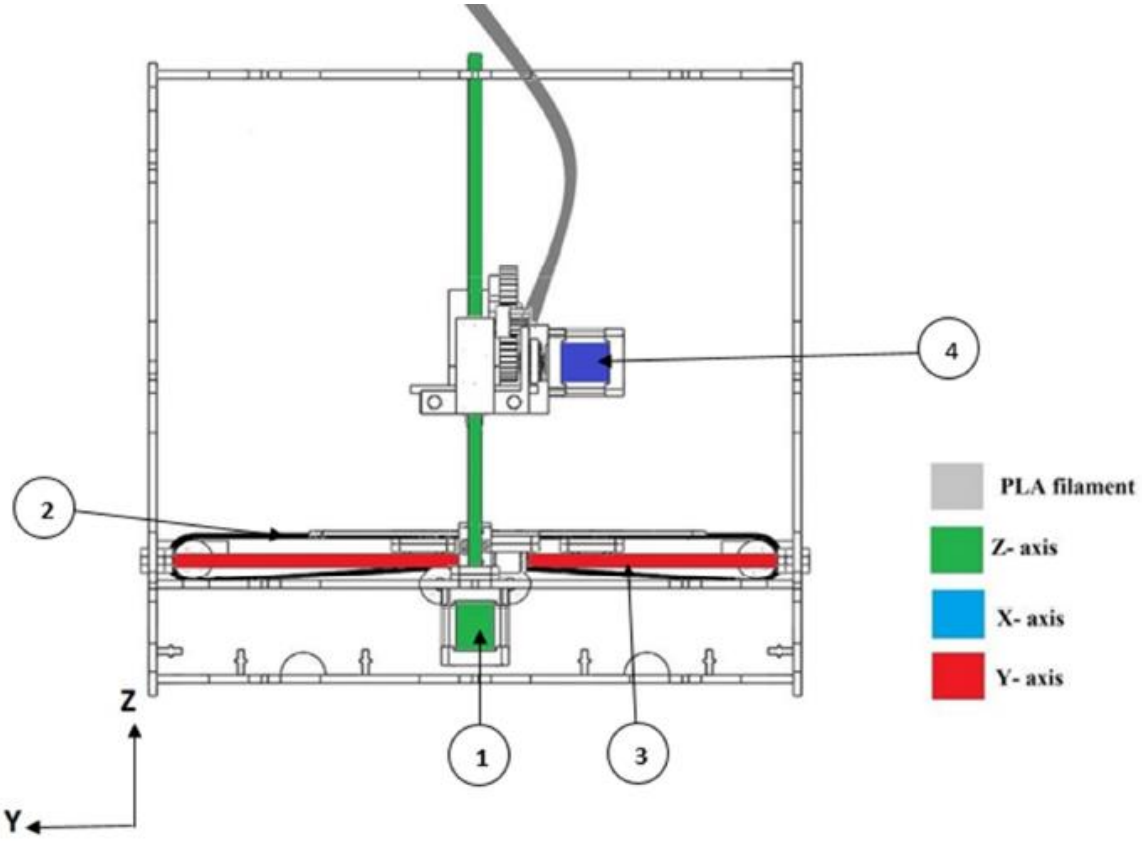
More detailed diagrams

- Front view



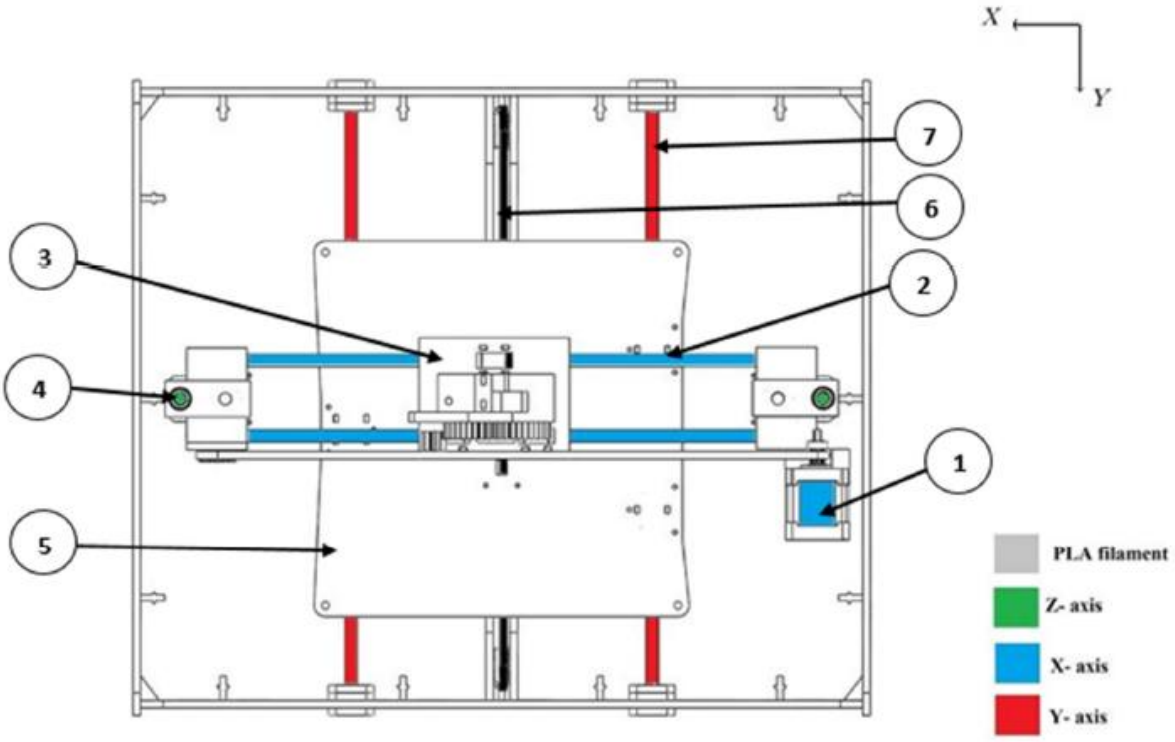
1. X axis motor
2. Y axis motor
3. Z axis motors
4. Extruder
5. Timing belt
6. 5/8 mm coupler
7. Y axis linear bearing
8. Z axis smooth rods
9. Z axis threaded rods
10. Nozzle

- **Side view**



1. Y axis motor
2. Timing belt
3. Y axis smooth rod
4. Extruder motor

- **Front view**



1. X axis motor
2. X axis smooth rod
3. Extruder
4. Z axis linear bearings
5. Print bed
6. Y axis timing belt
7. Y axis smooth rod

Software downloading

- Slic3r

Download “Slic3r” software from this link (it is an open source software): http://slic3r.org/download. Notice that when you setup “Slic3r” and open it for the first time. The configuration wizard asks a series of questions and creates a configuration for Slic3r to start with.

For this printer choose these values respectively:
- Firmware Type: “RepRap (Marlin/Sprinter)”
- Bed size: x 200 y 200 mm
- Nozzle Diameter: .45 mm
- Filament Diameter: 2.85 mm
- Extrusion Temperature: 200 C
- Bed Temperature 80 C

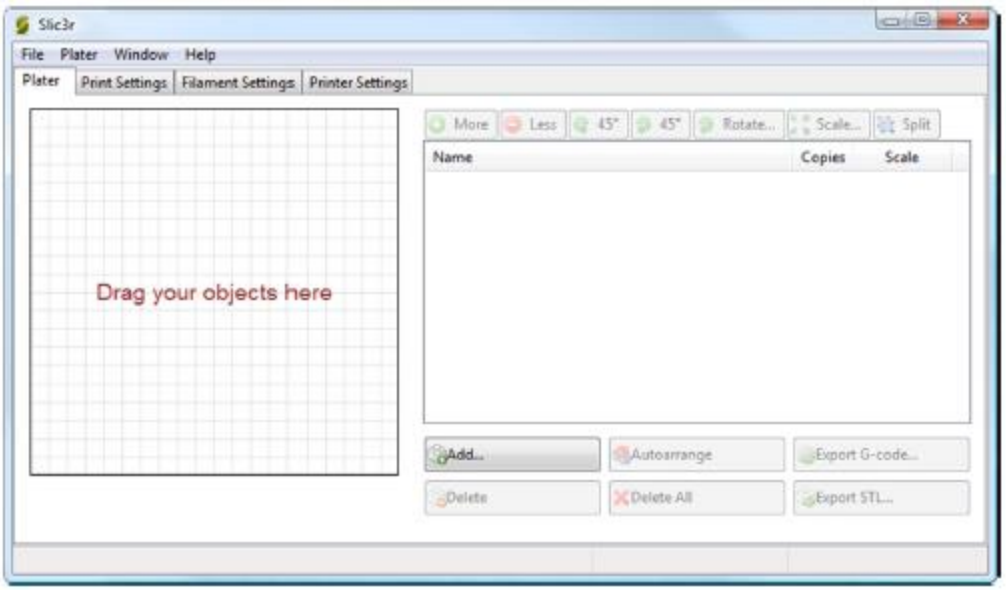
Notice that you can change these settings from inside the software at any time (if you see that the temperature should be higher or lower for example).

- Pronterface

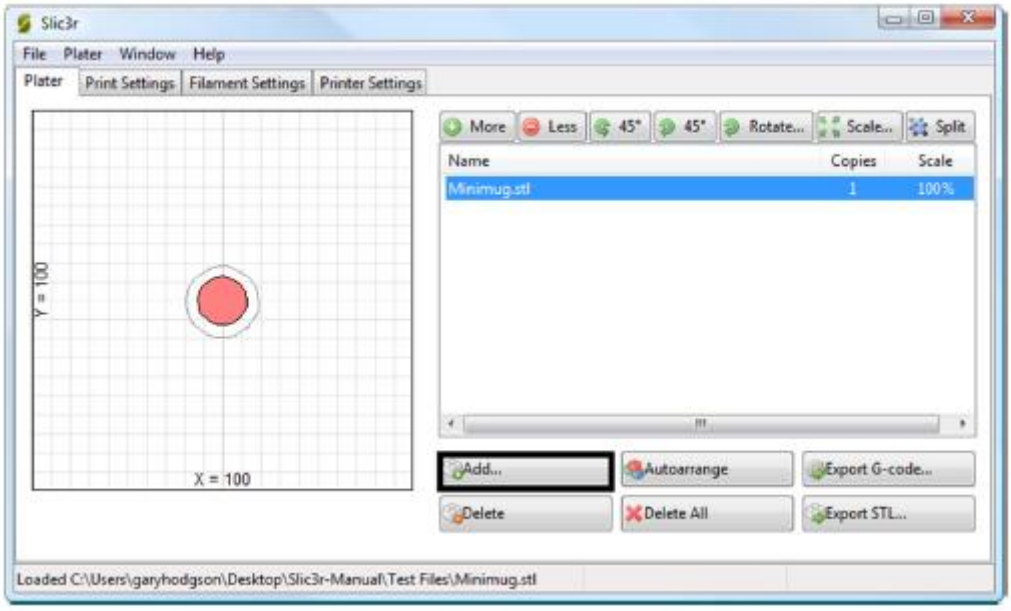
Download the latest version of “Pronterface” software from this link (it is an open source software): http://koti.kapsi.fi/~kliment/printrun/ .

Printing Procedure

1. Connect the printer to the computer using USB 2.0 cable.
2. Prepare you printer by:
_ Ensure that the print bed is clean.
_ Ensure that the print bed is flat, if not adjust it using four wing nuts.
_ Manually adjust the position of Z-axis limit switch to satisfy a suitable distance between the nozzle and the print bed (check it by pass a thin piece of paper between them).
3. Prepare 3D model in ".stl" file format you can download it from an online websites, such as "Thingiverse" or "GrabCAD", or create it using any CAD program such as "SolidWorks" or "SketchUp".
4. Open **"Sli3re"** software

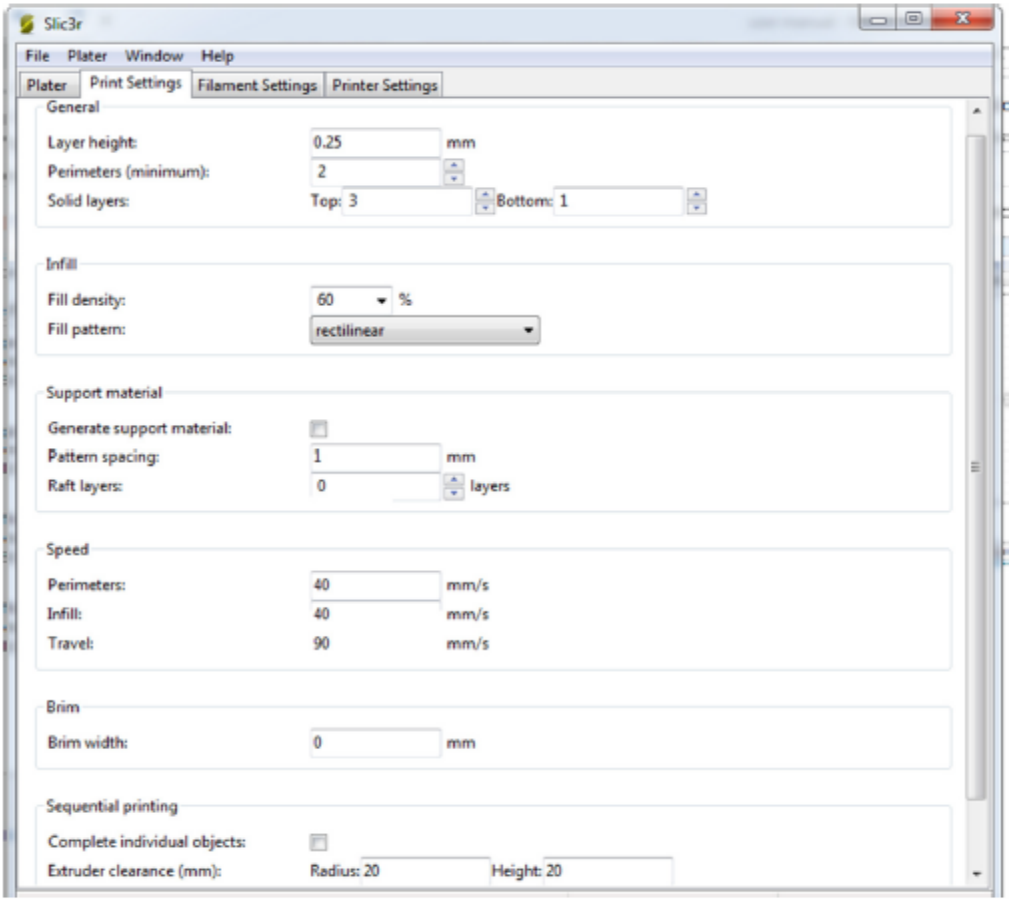


5. Load the 3D model to the Plater window from "Add" icon

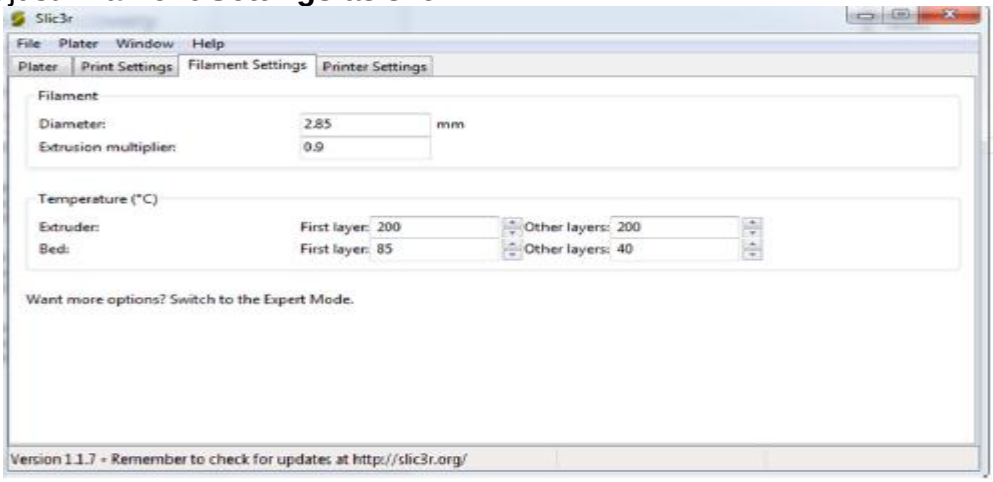


6. Adjust **Print Settings** as shown

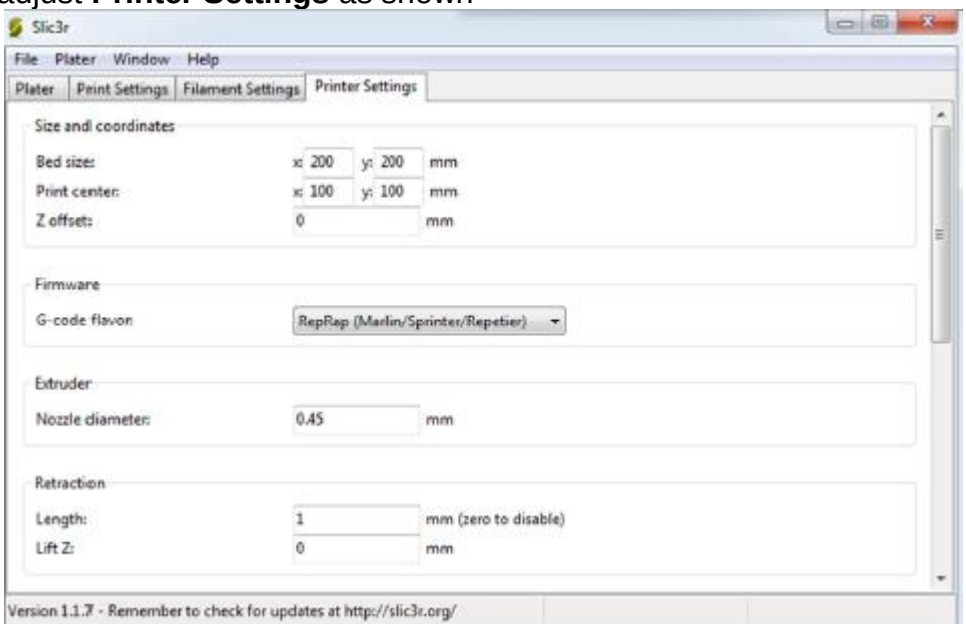
(The majority of these settings is chosen and tested by trial and error, so if it is needed you can change them to achieve better results, and this is the same for settings shown in next steps except the physical setting that relates to the dimensions of the printer and the filament).



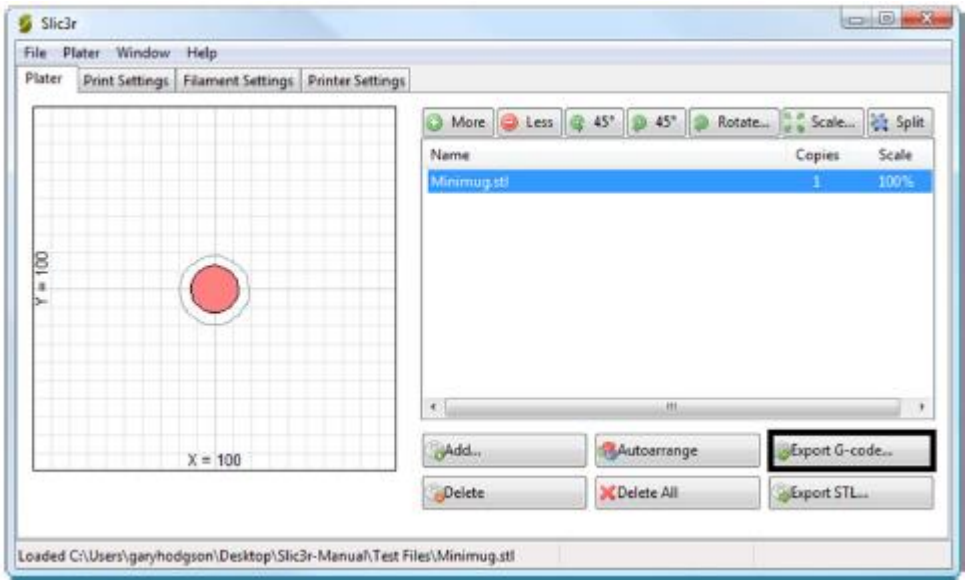
7.Also adjust **Filament Settings** as shown



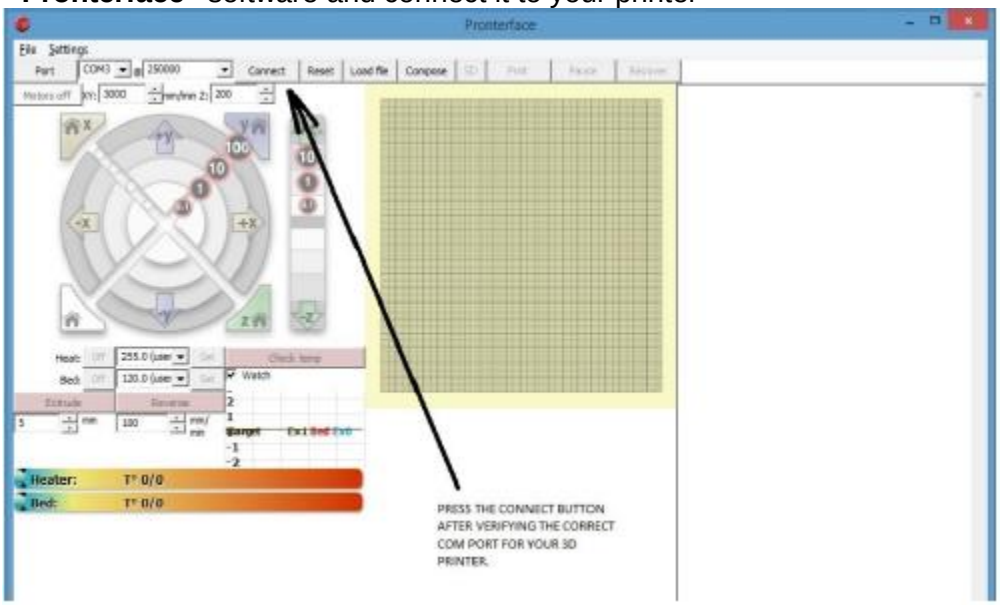
8. Finally adjust **Printer Settings** as shown



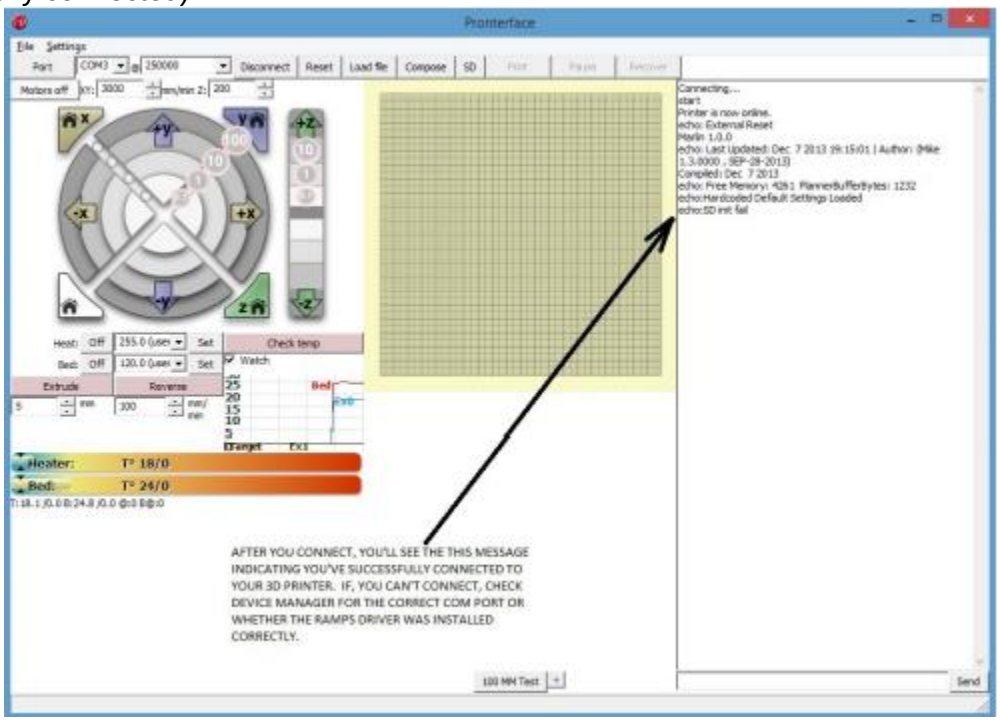
9. Export G-code



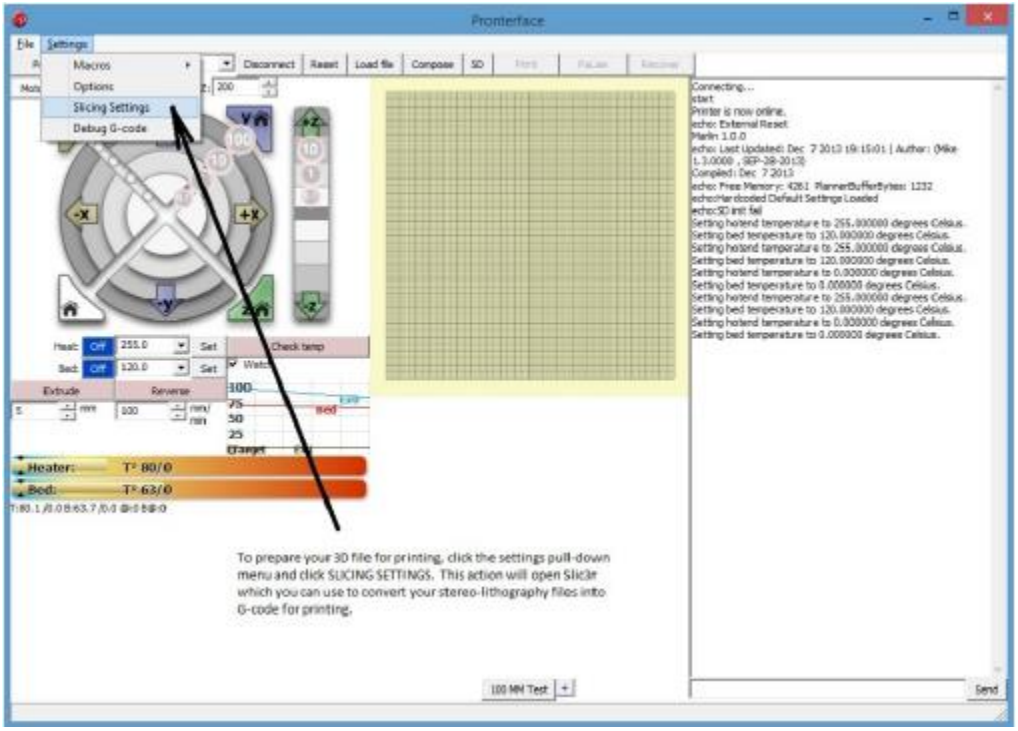
10. Open **“Pronterface”** software and connect it to your printer



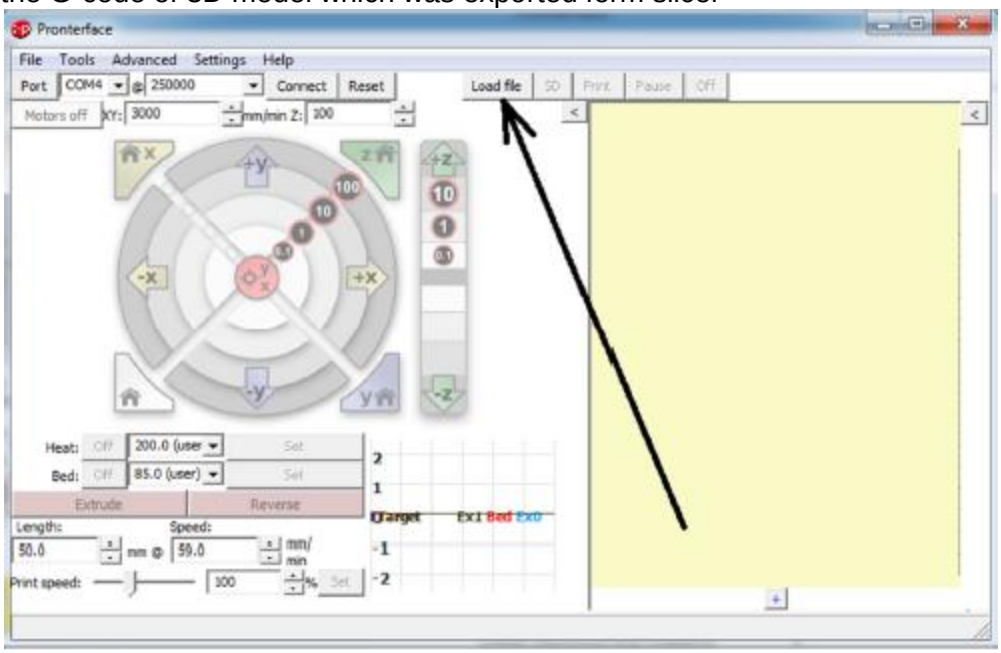
(You will see a message in the right column of Pronterface indicating that the printer has successfully connected)



11. From **settings list** Adjust settings as the same of “slic3r” settings (not as numbers shown in figure)



12. Load the G-code of 3D model which was exported form slicer



13. Click print icon to start printing (it will be enabled if the printer is connected and G-code is loaded)