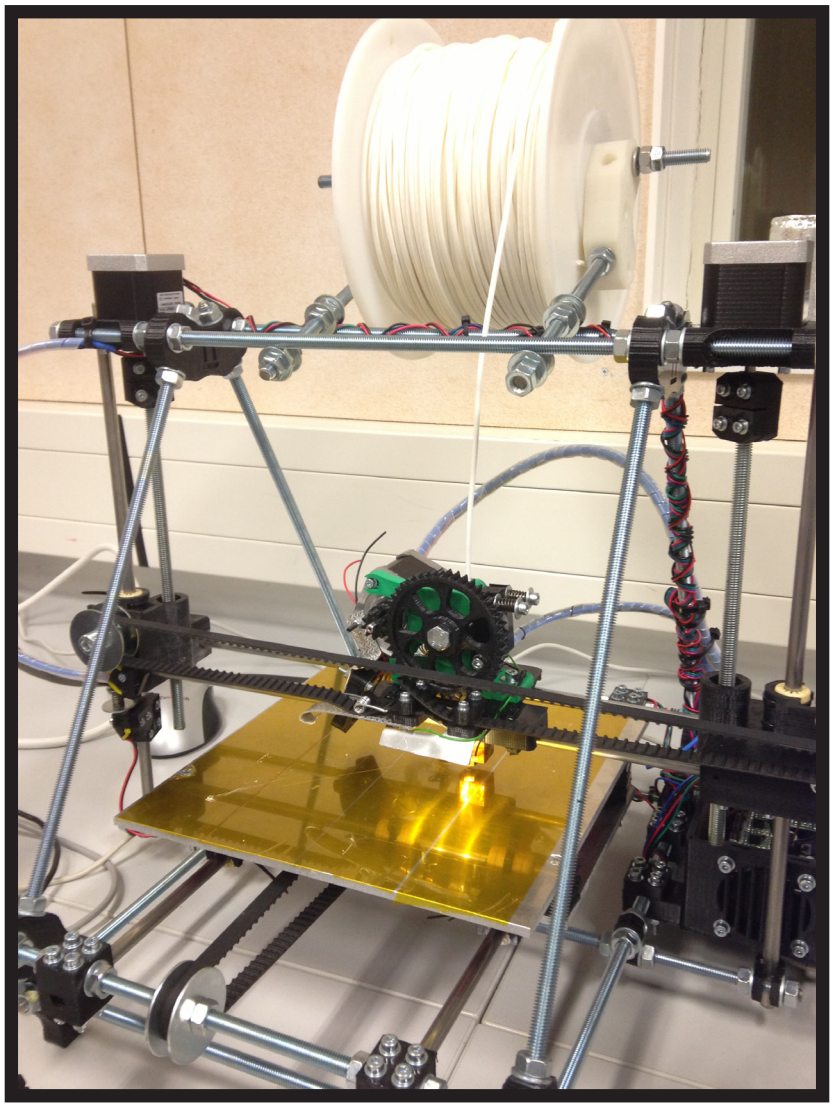
Manual
3D Printer PRUSA



This manual gives you an overview and short introduction.

Please read these instructions carefully and follow them as explained. This will guarantee that your prodccts look and feel great and also prevent the machines
from damage.

If you encounter a problem with the machines, do not try to fix it yourself, please get in contact with one of the fablab managers.

Contacts

Dr.-Ing. Dennis Krannich
krannich@tzi.de
0421-218-64384

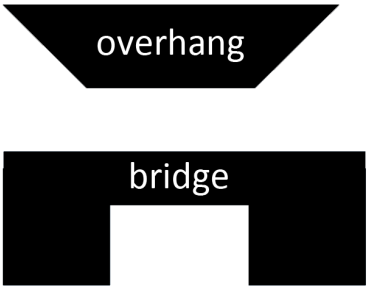
Jennifer Boldt
boldti@tzi.de

Rules!

- Start printing only when temperatures are reached!
  Otherwise: damage to the extruder
- Risk of burns from hot parts:
  - Bed-plate (depending on settings) is hot
  - **Extruder** is very hot! Do **not touch**!
- Turn off the printer: using the switch on the power supply (Shutting down the PC is not enough!)
  - if the printer is not required any more
  - if the printer makes nonsense (strange noise/not moving in home position/etc.)
    - if the printer is not moving in home position, move the X-axis by hand (only when engines are off)
- Move the axes manually
  - **Z-axis** (height) do not **move** by hand, **only via software** (eg. Z +10)
  - X-axis and Y-axis only move by hand when engines are off
- **Do not change pre-setting profiles on the PC!**

File Format

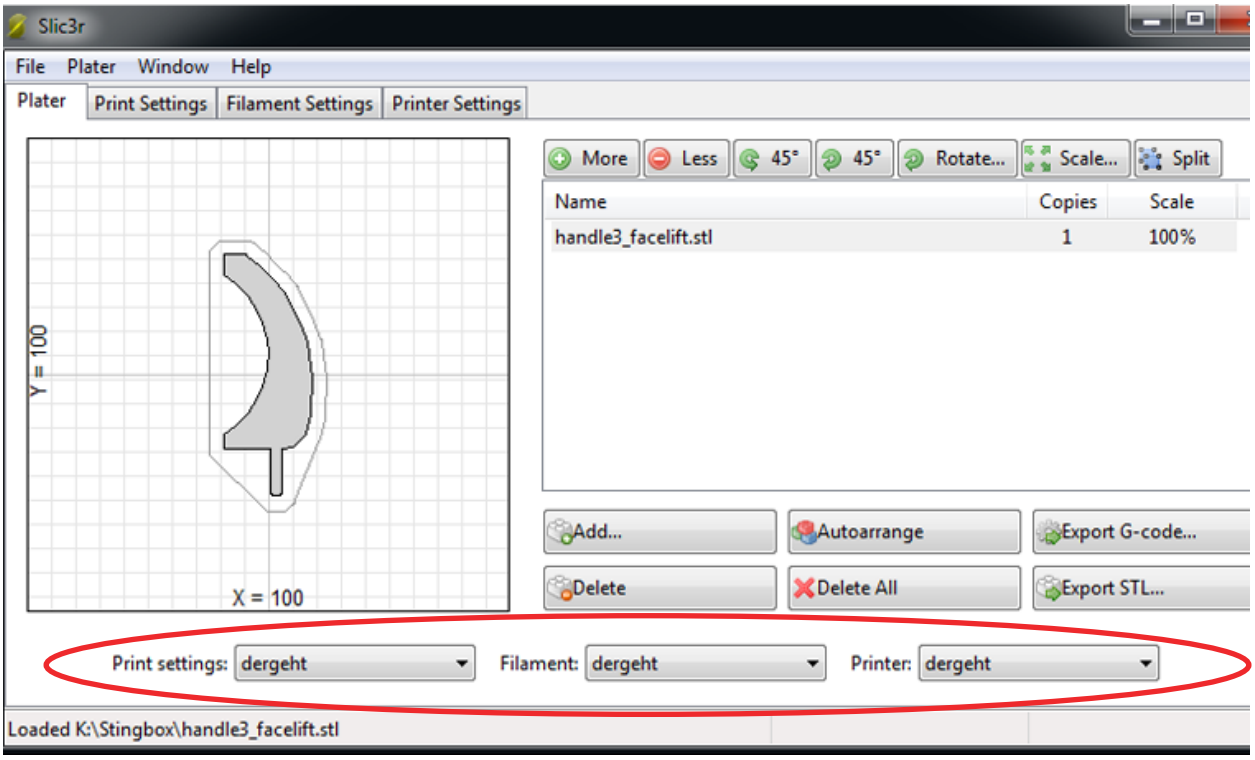
- STL file, unit: millimeters
  - Create a 3D object and export your file as STL file (3D programs: Blender, Sketchup, 3DS Max, OpenSCAD, etc.)
- Rules for your 3D objects:
  - Object must be completely closed
  - Not hollow inside
  - Overhangs should not be too large (<45°)
  - Bridges should not be too far (<5mm)



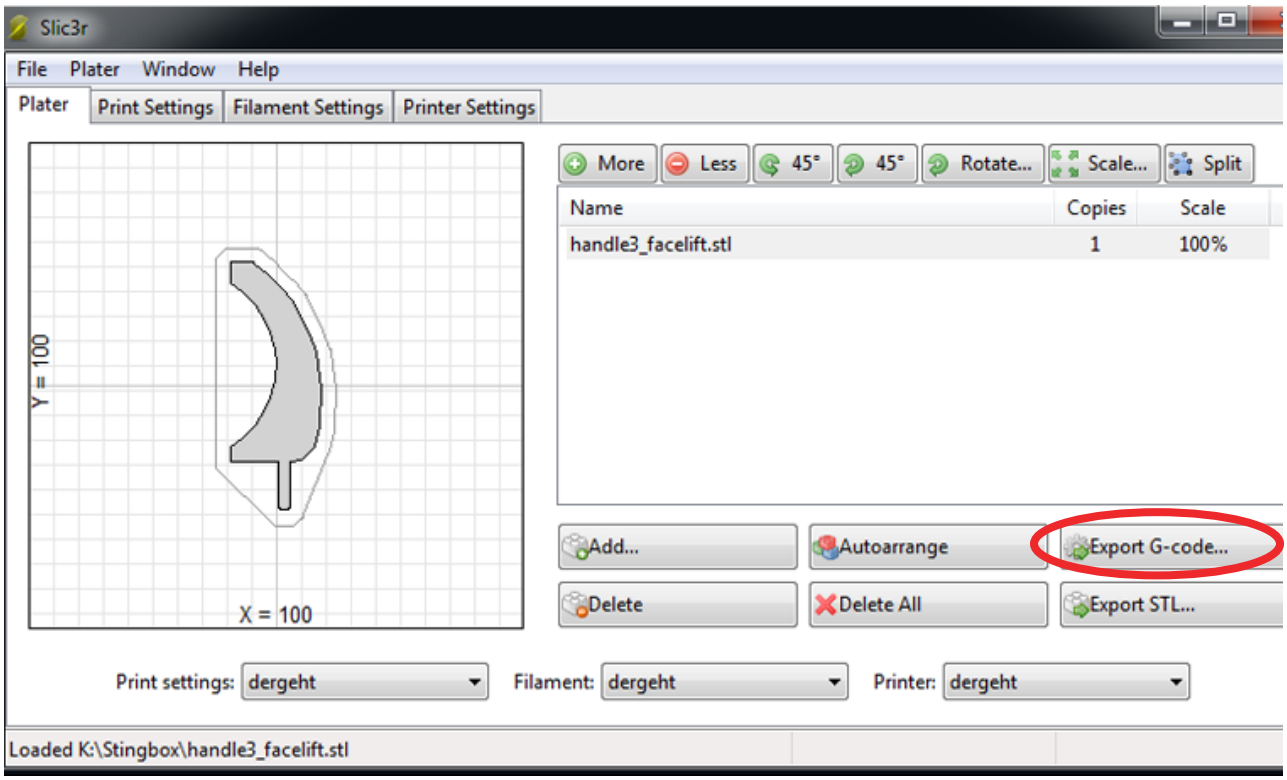
- Maximum dimensions X 180mm, Y 180mm, Z 130mm

Convert 3D model (STL file) into gcode

- Start „Slic3r“ (on desktop)
- Load your STL file via „Drag & Drop“
- Do not change pre-settings (Printer settings: „dergeht“/Filament:“dergeht“/Printer:“dergeht“)



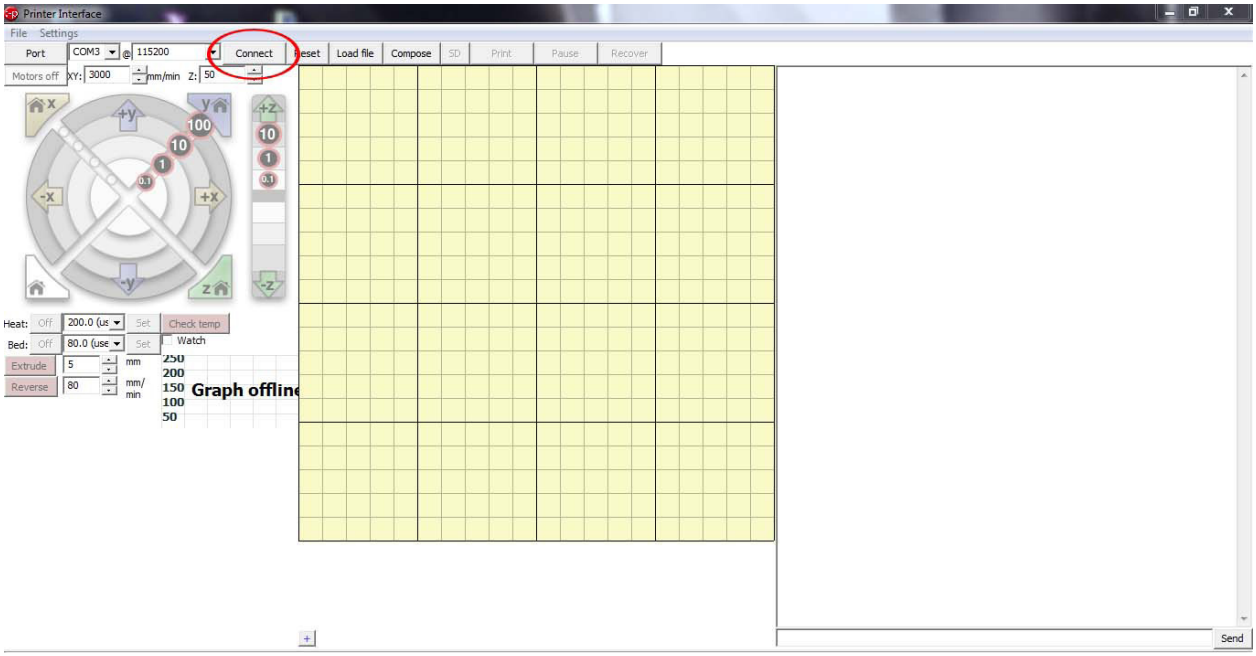
- Click „Export G-code...“



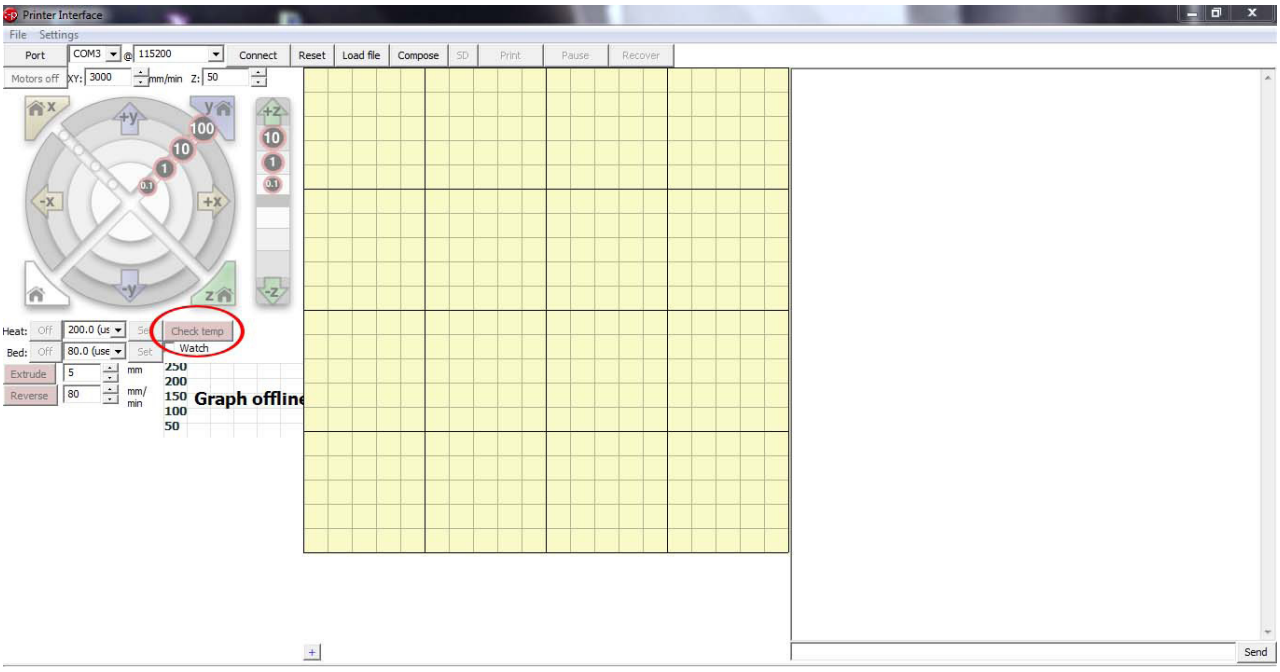
- Wait...finished

Printer Control

- Printer on (switch at the power supply)
- Start “Pronterface” (on desktop)
- Click “Connect” (wait -> “Printer is online”)

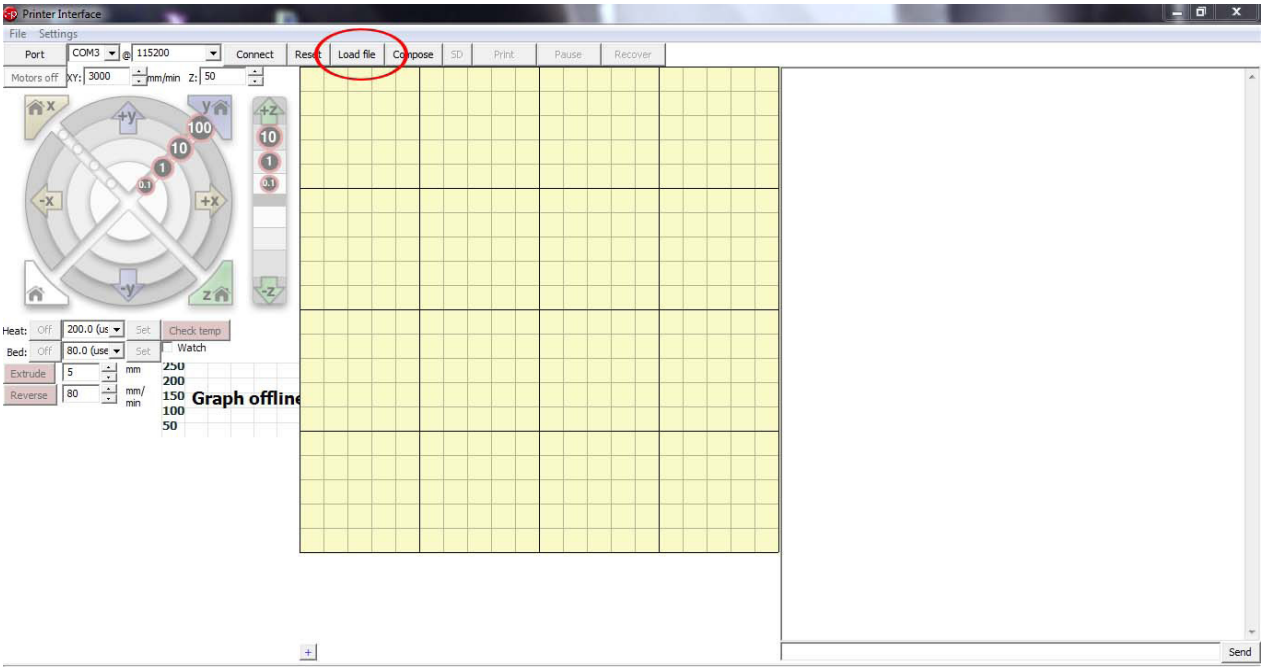


- Click “Check temp”

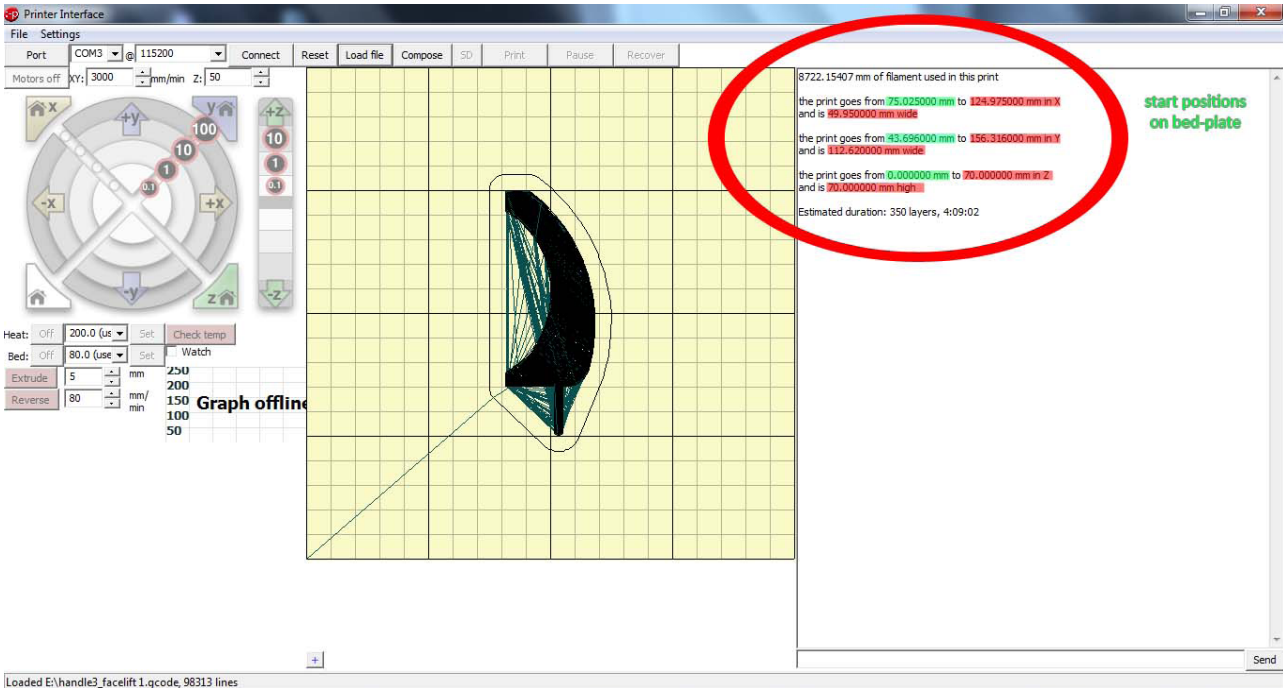


Printing

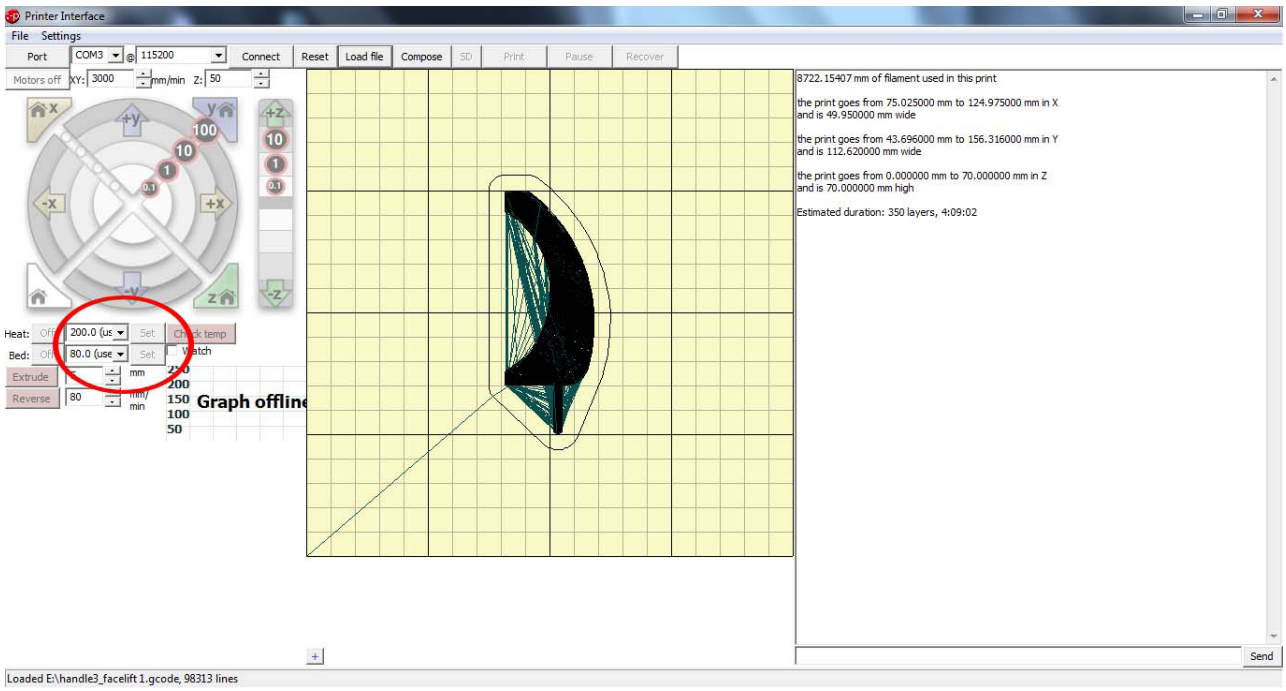
1. Load your xyz.gcode



2. The boundary points of the object are displayed: X/Y/Z goes from ... to ... mm
   a. check the size -> not too big? (max 180mm x 180mm x 130mm)



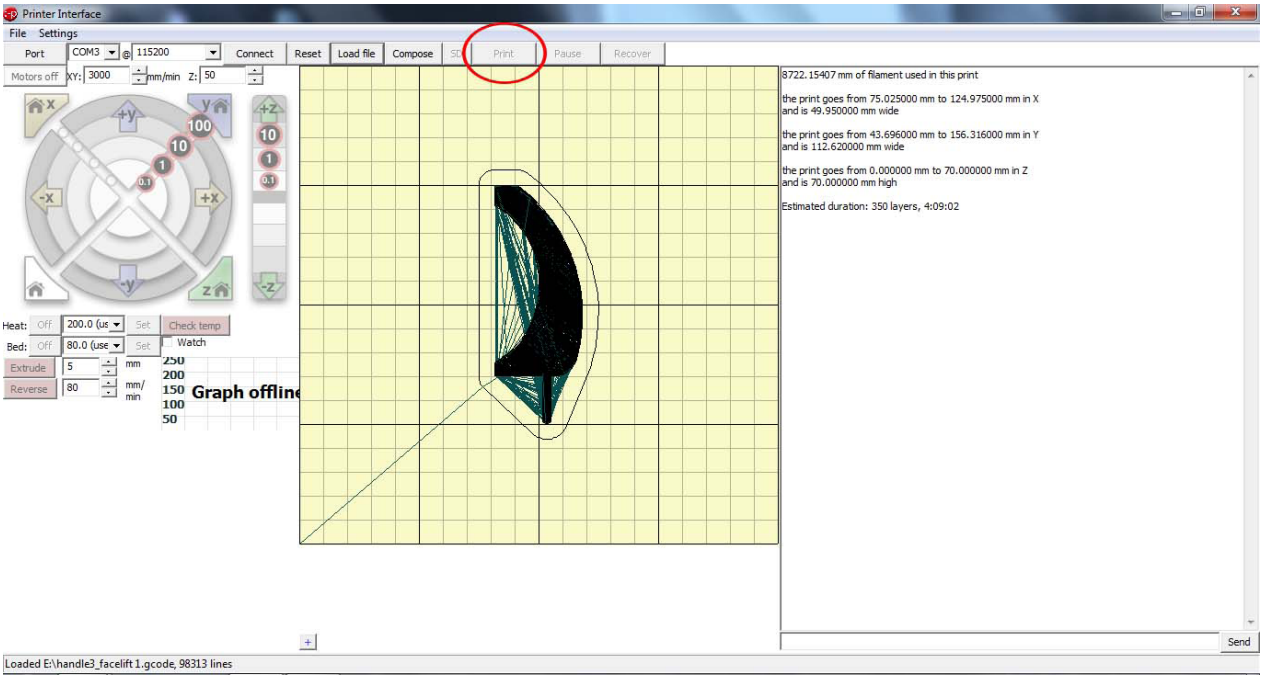
3. Set temperatures (Heater: 200 °C and Bed: 80°C)



4. If necessary: clean the bed-plate
   a. only with hands or scraper carefully (Attention: Hot!) (no liquid, oil or something else)
   b. if necessary move the printer head (Extruder) upwards (several times Z+10)

5. Check temperatures: Heater 200 °C and Bed 80° C

6. Click “print” (you see a countdown from 9 to 0)



7. Printing is finished: wait a few minutes until the object is cold -> take object from the bed-plate

8. Clean the bed-plate (see 4.)

9. Disconnect printer

10. Close “Pronterface”

11. Turn off the printer: using the switch on the power supply!