

The information on this manual was written with software running on Microsoft Windows 8.1 but the same software can be found for MAC and Linux and the same instructions may work on other operation systems.

Software setup

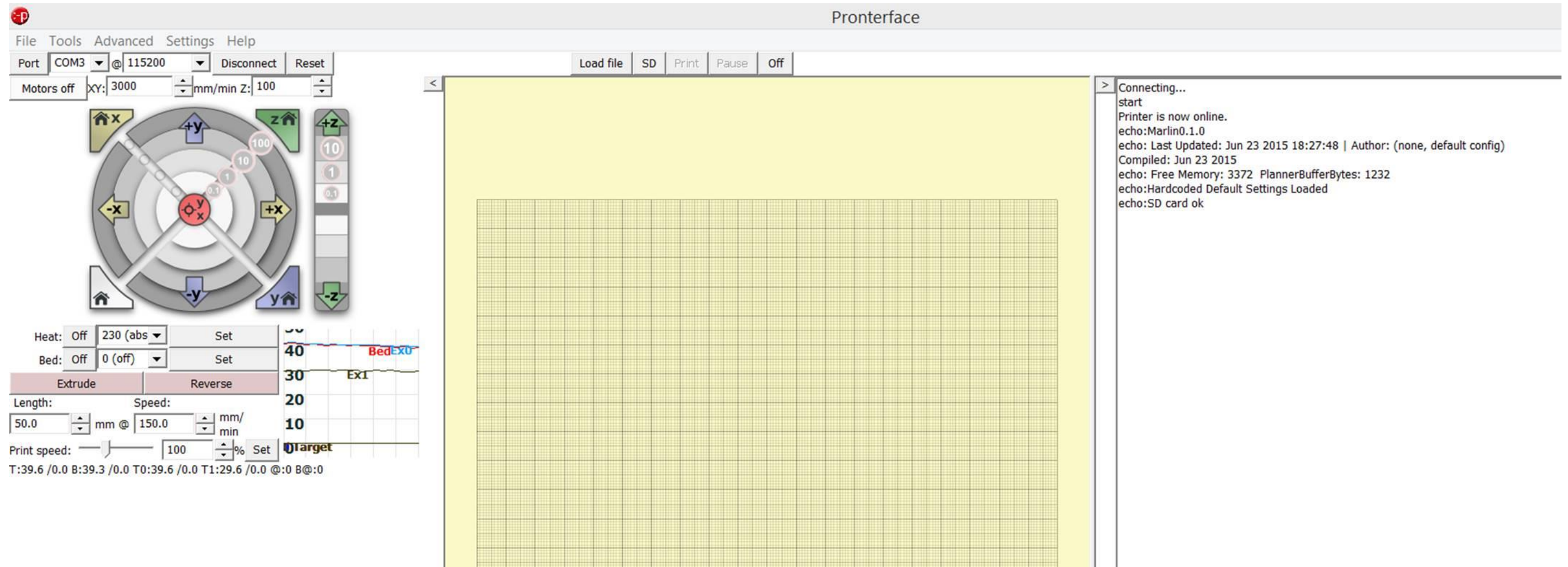
In order to be able to control helloBEEprusa and print, you need to use an OpenSource software that is developed by the 3D printing community.

Pronterface

In order to help you calibrate the print bed, load/unload filament and do other operations, you can use the Pronterface software. Please download it from this link - version "Printrun-Win-Slic3r-03Feb2015":

<http://koti.kapsi.fi/~kliment/printrun/>

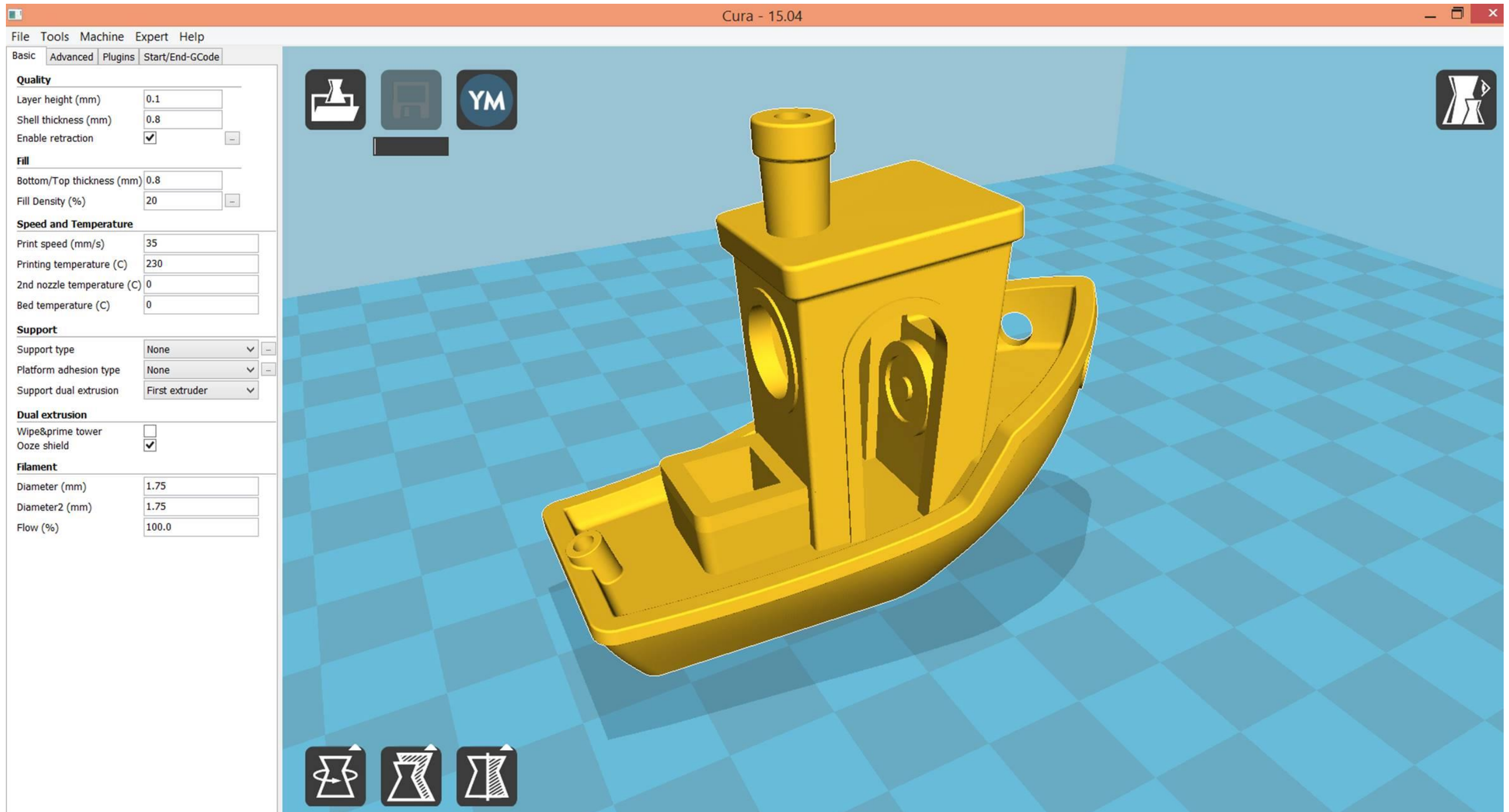
Pronterface screenshot:



Cura 15.04

In order to generate the G-code for printing, you can use the Cura 15.04. Please download it from this link - version "Version: 15.04": <https://ultimaker.com/en/cura-software/list>

Cura 15.04 screenshot:



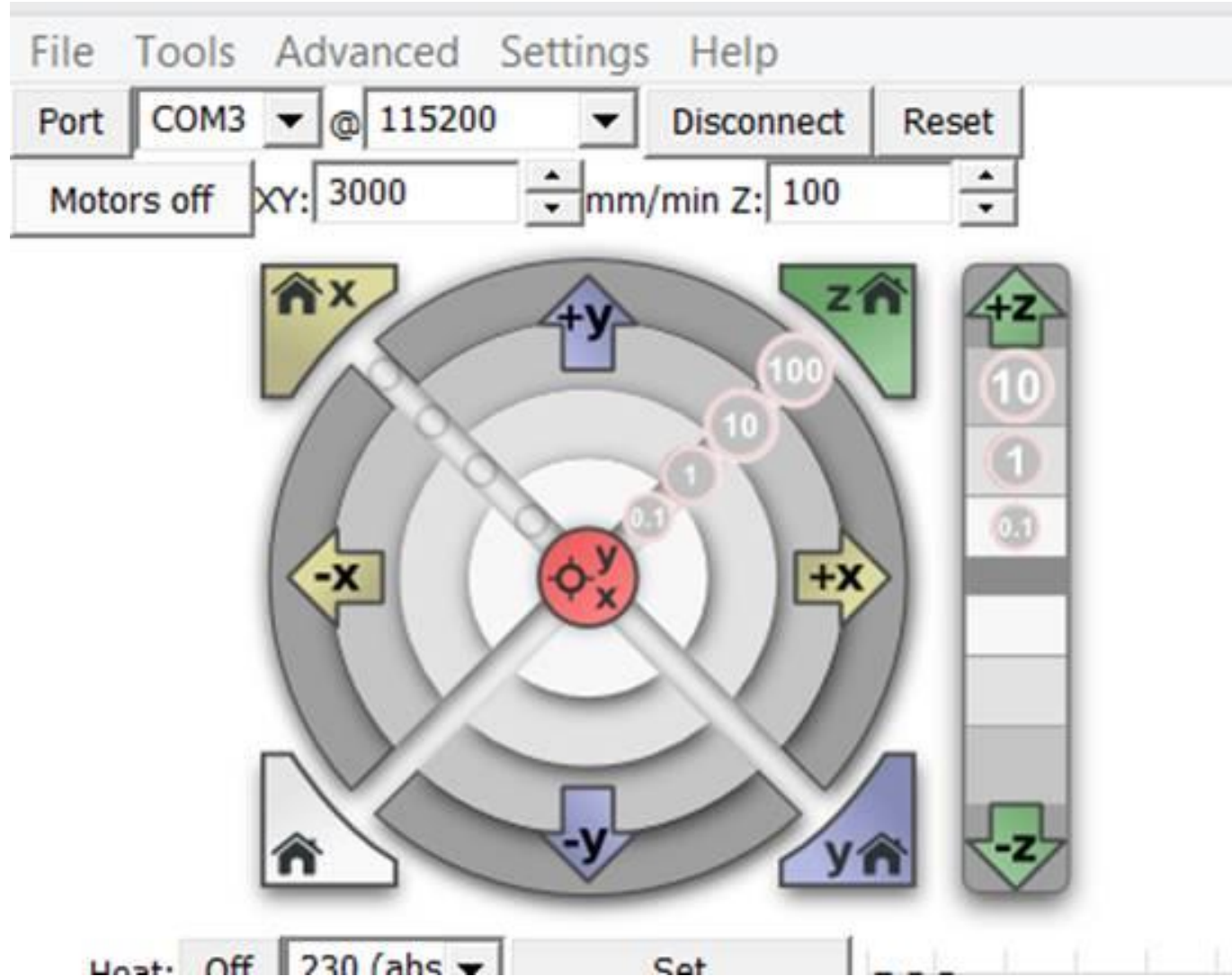
Preparing for first print

In order to be able to print you need to first calibrate your bed and load/unload the filament.

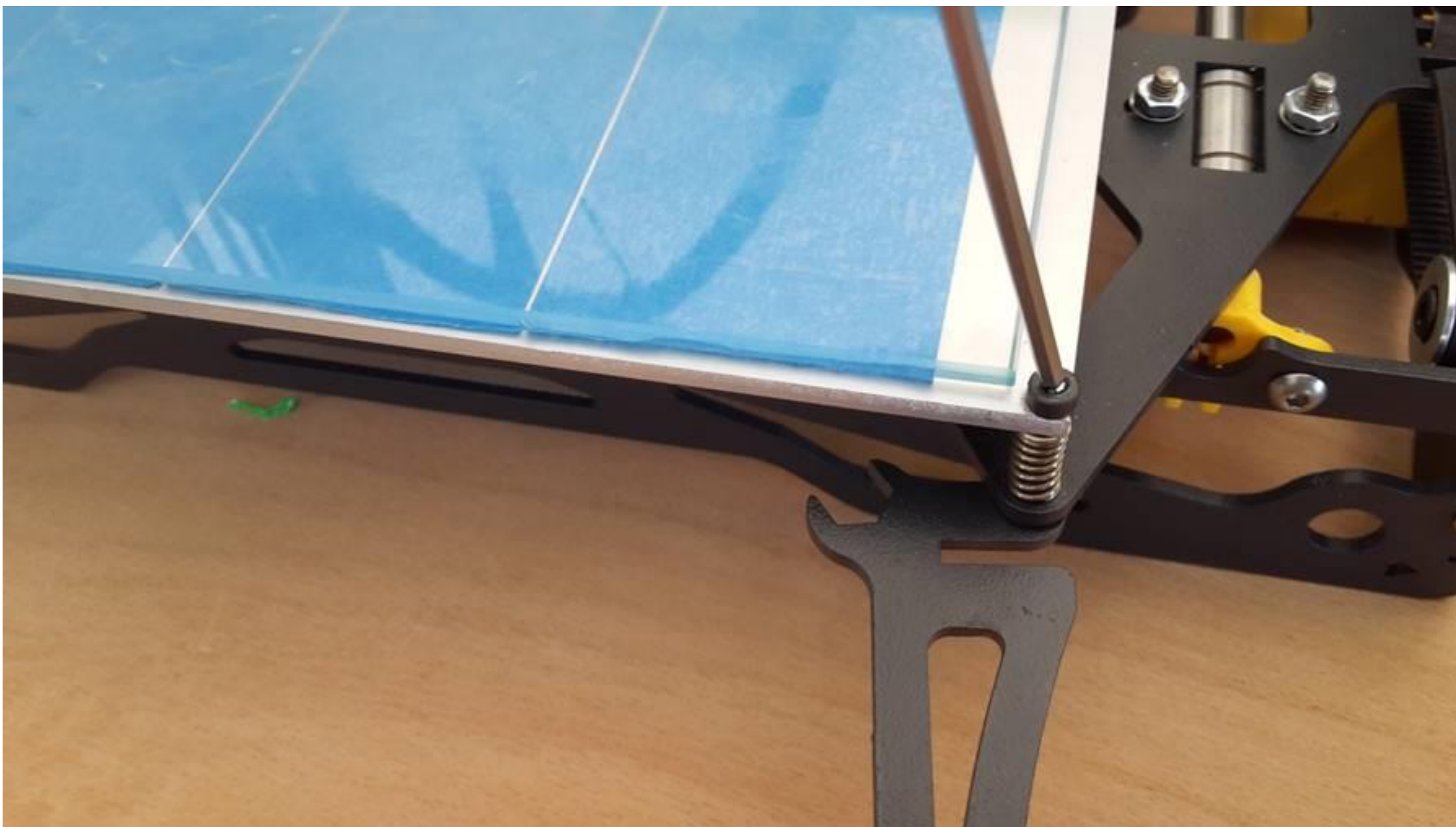
Bed calibration

You need to evenly calibrate the bed so the first layer of the print can adhere well to the bed, otherwise it may ruin your print.

Before starting to level the bed, you can jog the extruder and the bed using the jog buttons on Pronterface. You can move the Z axis in a way that the nozzle stays a bit closer to the bed and is essential for the following steps.

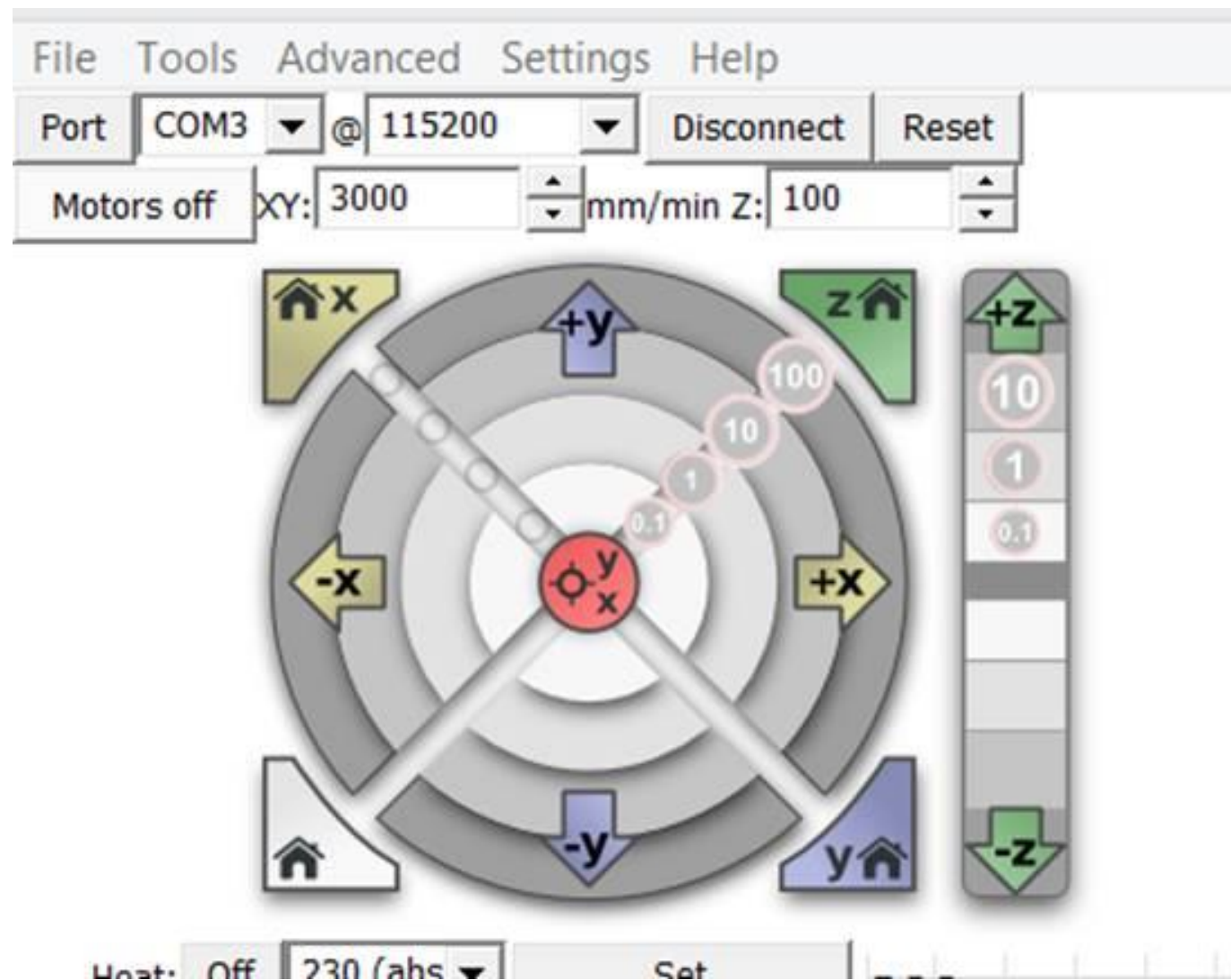


Now use the 3 screws to level the bed. You need to do it keeping the same distance from the nozzle on all points of the bed.



Next step is to adjust the home Z axis so the nozzle can stay at a distance of about 0.2mm from the bed. You can use a paper sheet as reference for the needed distance.

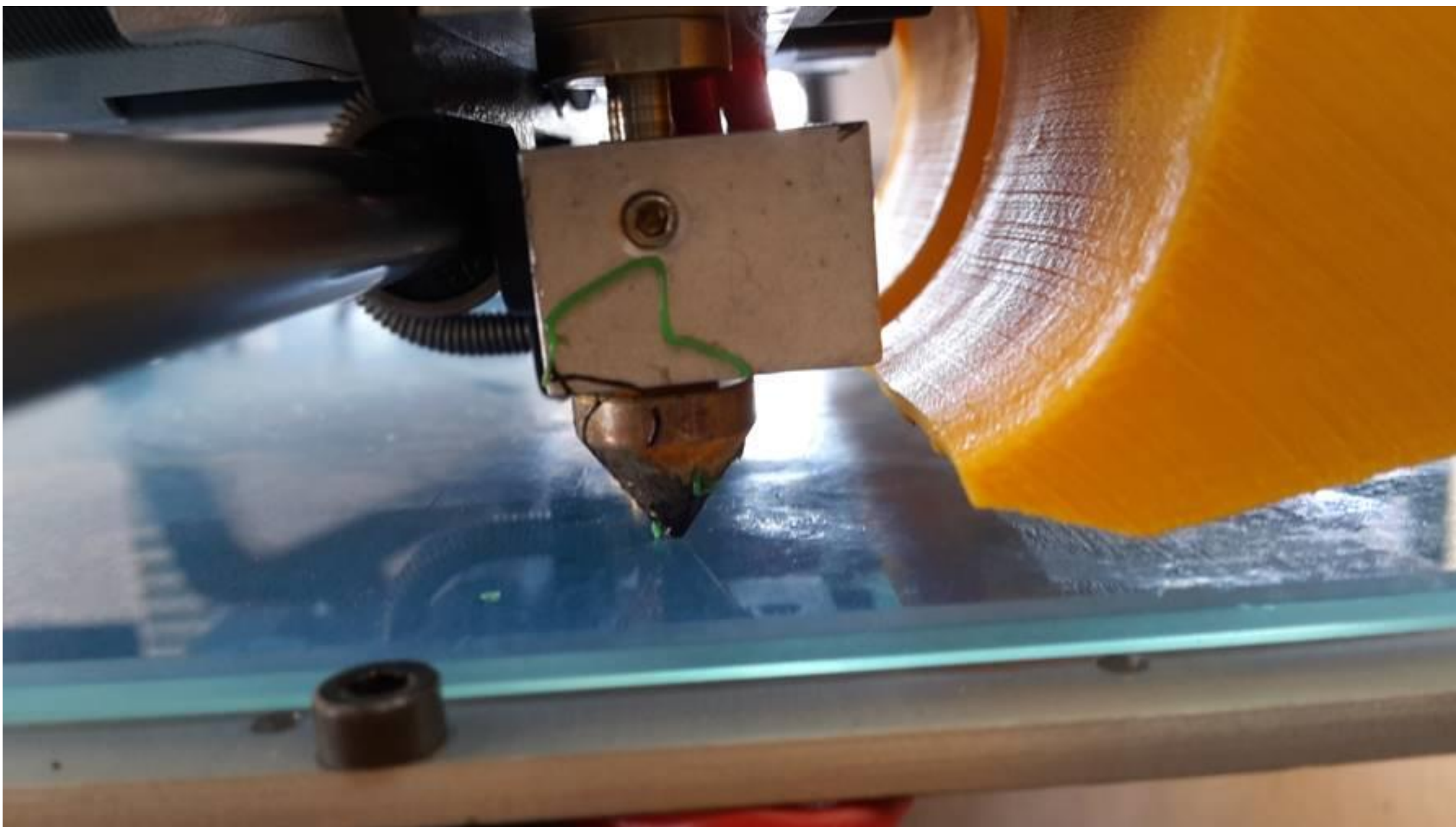
Use the Pronterface jog panel to home the Z axis:



Now adjust the placement of Z endstop so when the home of Z axis ends, the nozzle stays at a distance of about 0.2mm from the bed.

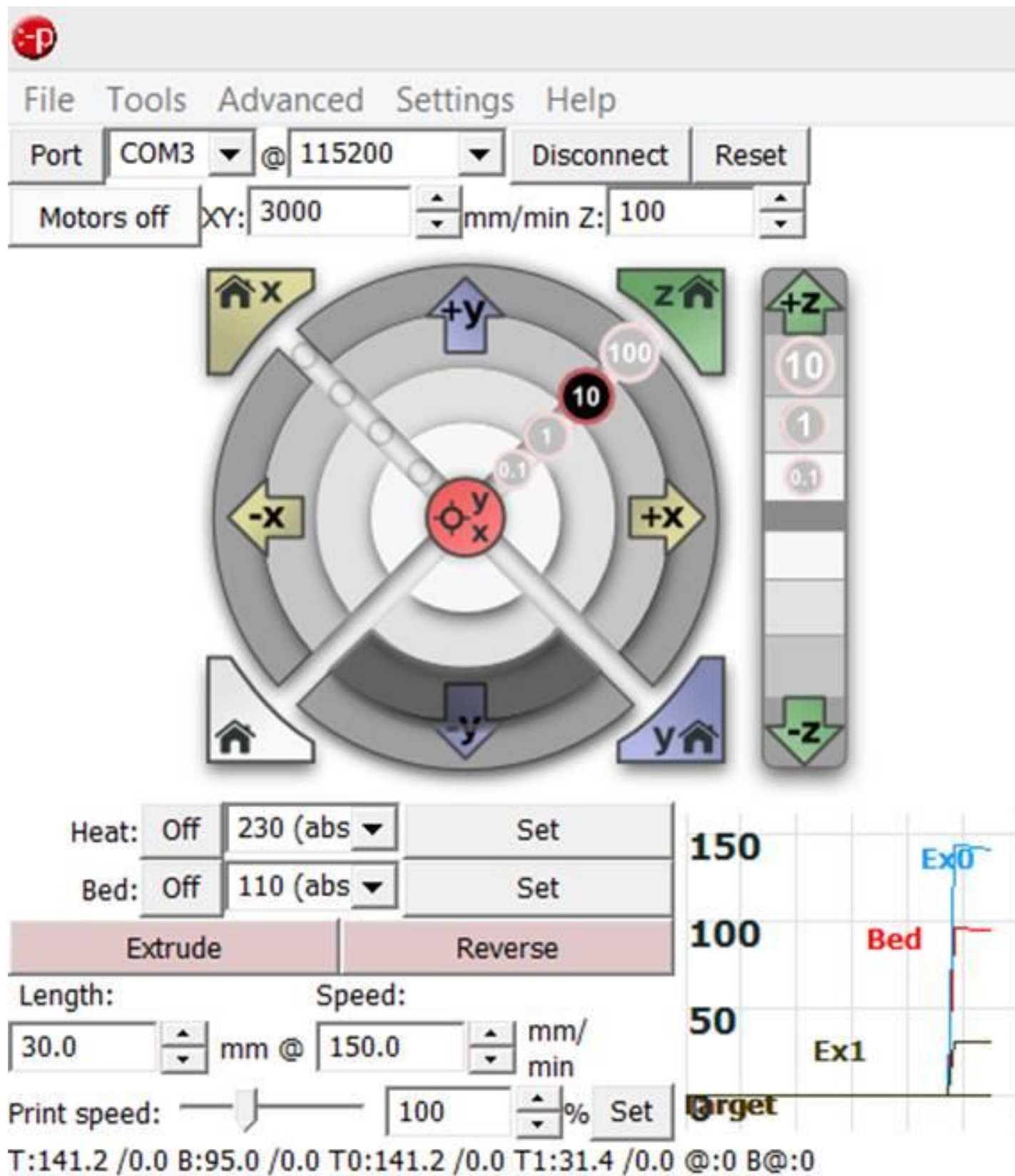


Example of the nozzle staying at about 0.2mm from the bed:



[Load/unload the filament](#)

You can load/unload filament using the Pronterface control panel:



To load, first you need to set the extruder heating temperature and wait for the temperature to reach that value. You can look at the graph to verify.

When the extruder reaches the temperature, then you can click on the Extrude button and then you can insert the filament on the extruder.

To unload, you also need to set the same extruder heating temperature. Afterwards, just click the Reverse button and gently pull the filament from the extruder.

If you want to control the second extruder (ex 1), send the command "T1" on the Pronterface console in order to control the second extruder. Send the command "T0" to control again the first extruder.

First print

For our first print we will be using the free 3DBenchy 3D model - you can download it here: <http://www.3dbenchy.com/>

We will also configure Cura 15.04 to export the G-code. Finally we will save the G-code file on the SDCard and print it.

Add helloBEEprusa as a new machine on Cura 15.04

Go to "File --> Machine settings" and click on "Add new machine".

Follow the next screenshots and use the same options.

Select your machine

What kind of machine do you have:

- ☐ Ultimaker2
- ☐ Ultimaker2extended
- ☐ Ultimaker2go
- ☐ Ultimaker Original
- ☐ Ultimaker Original+
- ☐ Printrbot
- ☐ Lulzbot TAZ
- ☐ Lulzbot Mini
- ☒ Other (Ex: RepRap, MakerBot, Witbox)

The collection of anonymous usage information helps with the continued improvement of Cura.

This does NOT submit your models online nor gathers any privacy related information.

Submit anonymous usage information: ☒

For full details see: <http://wiki.ultimaker.com/Cura:stats>

Other machine information

The following pre-defined machine profiles are available

Note that these profiles are not guaranteed to give good results, or work at all. Extra tweaks might be required.

If you find issues with the predefined profiles, or want an extra profile.

Please report it at the github issue tracker.

- ☐ BFB
- ☐ DeltaBot
- ☐ Hephestos
- ☐ Hephestos_XL
- ☐ MakerBotReplicator
- ☐ Mendel
- ☐ Ord
- ☐ Prusa Mendel i3
- ☐ Rigid3D
- ☐ RigidBot
- ☐ RigidBotBig
- ☐ Witbox
- ☐ Zone3d Printer
- ☐ julia
- ☐ katihal
- ☐ punchtec Connect XL

☒ Custom...

< Back

Next >

Cancel

Custom RepRap information

RepRap machines can be vastly different, so here you can set your own settings.

Be sure to review the default profile before running it on your machine.

If you like a default profile for your machine added,
then make an issue on github.

You will have to manually install Marlin or Sprinter firmware.

Machine name	<input type="text" value="helloBEEprusa"/>
Machine width X (mm)	<input type="text" value="250"/>
Machine depth Y (mm)	<input type="text" value="200"/>
Machine height Z (mm)	<input type="text" value="200"/>
Nozzle size (mm)	<input type="text" value="0.4"/>
Heated bed	<input checked="" type="checkbox"/>
Bed center is 0,0,0 (RoStock)	<input type="checkbox"/>

< Back

Finish

Cancel

Access to "Machine --> Machine settings..." and compare with the following screenshot:

Machine settings

Ultimaker2 Beethefirst Hellobeeprusa

Machine settings

E-Steps per 1mm filament 0

Maximum width (mm) 250

Maximum depth (mm) 200

Maximum height (mm) 200

Extruder count 2

Heated bed ☒

Machine center 0,0 ☐

Build area shape Square

GCode Flavor RepRap (Marlin/Sprinter)

Printer head size

Head size towards X min (mm) 0

Head size towards Y min (mm) 0

Head size towards X max (mm) 0

Head size towards Y max (mm) 0

Printer gantry height (mm) 0

Communication settings

Serial port AUTO

Baudrate 115200

Extruder 2

Offset X 0

Offset Y 0

Ok Add new machine Remove machine Change machine name

Access to "Expert --> Open expert settings..." and compare with the following screenshot:

Expert config

**Dual extrusion**Wipe&prime tower volume per layer (mm3) **Retraction**Minimum travel (mm) Enable combing Minimal extrusion before retracting (mm) Z hop when retracting (mm) **Skirt**Line count Start distance (mm) Minimal length (mm) **Cool**Fan full on at height (mm) Fan speed min (%) Fan speed max (%) Minimum speed (mm/s) Cool head lift ☐**Infill**Solid infill top ☒Solid infill bottom ☒Infill overlap (%) Infill prints after perimeters ☐**Support**Structure type Overhang angle for support (deg) Fill amount (%) Distance X/Y (mm) Distance Z (mm) **Black Magic**Spiralize the outer contour ☐Only follow mesh surface ☐**Brim**Brim line amount **Raft**Extra margin (mm) Line spacing (mm) Base thickness (mm) Base line width (mm) Interface thickness (mm) Interface line width (mm) Airgap First Layer Airgap Surface layers Surface layer thickness (mm) Surface layer line width (mm) **Fix horrible**Combine everything (Type-A) ☐Combine everything (Type-B) ☐Keep open faces ☐Extensive stitching ☐

Ok

Configure settings and export GCode

Follow these screenshots and use the same options.

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/End-GCode

Quality

Layer height (mm)

0.1

Shell thickness (mm)

0.8

Enable retraction

☒

...

Fill

Bottom/Top thickness (mm)

0.8

Fill Density (%)

20

...

Speed and Temperature

Print speed (mm/s)

35

Printing temperature (C)

240

2nd nozzle temperature (C)

240

Bed temperature (C)

100

Support

Support type

Touching buildplate

▼

...

Platform adhesion type

None

▼

...

Support dual extrusion

Second extruder

▼

Dual extrusion

Wipe&prime tower

☐

Ooze shield

☒

Filament

Diameter (mm)


1.75

Diameter2 (mm)

1.75

Flow (%)

100.0



3 hou

3.99 m

1.00

NOTE: for ABS filament, choose Printing temperature of about 240°C and Bed temperature of 100°C or more.
For PLA filament, choose Printing temperature of about 190°C and Bed temperature of about 60°C.

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/End-GCode

Machine

Nozzle size (mm)

0.4

Retraction

Speed (mm/s)

40.0

Distance (mm)

4.5

Dual extrusion switch amount (mm)

16.5

Quality

Initial layer thickness (mm)

0.3

Initial layer line width (%)

100

Cut off object bottom (mm)

0.0

Dual extrusion overlap (mm)

0.2

Speed

Travel speed (mm/s)

150.0

Bottom layer speed (mm/s)

20

Infill speed (mm/s)

0.0

Top/bottom speed (mm/s)

0.0

Outer shell speed (mm/s)

0.0

Inner shell speed (mm/s)

0.0

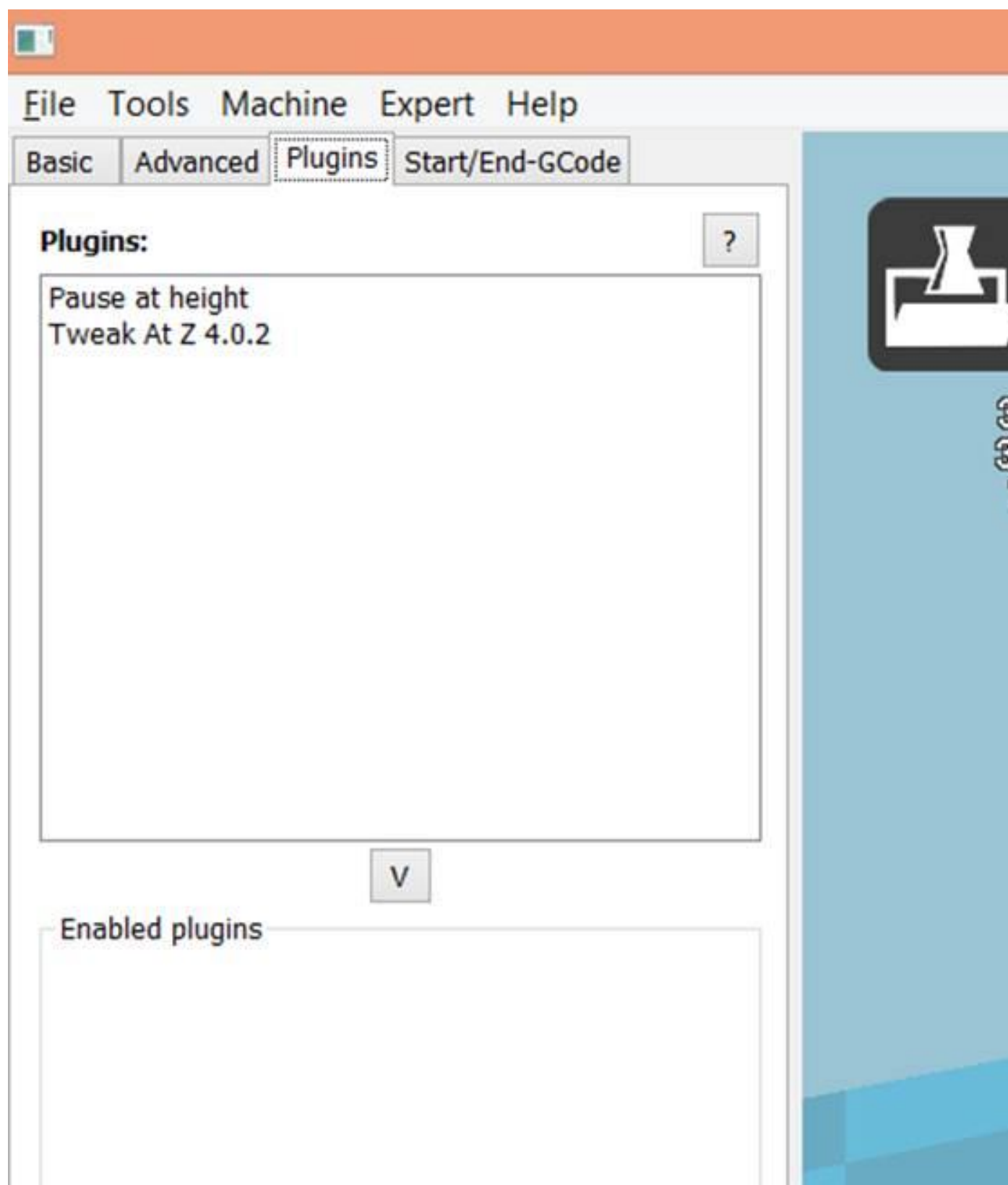
Cool

Minimal layer time (sec)

5

Enable cooling fan

☒





Use the following commands for start.gcode:

```
;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height} Walls: {wall_thickness} Fill: {fill_density}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament_weight}g
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to add your own bed temperature line
;M109 S{print_temperature} ;Uncomment to add your own temperature line
G21 ;metric values
G90 ;absolute positioning
M107 ;start with the fan off
G28 X0 Y0 ;move X/Y to min endstops
G28 Z0 ;move Z to min endstops
G1 Z15.0 F{travel_speed} ;move the platform down 15mm
G92 E0 ;zero the extruded length
G1 F200 E3 ;extrude 3mm of feed stock
G92 E0 ;zero the extruded length again
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...
```

Use the following commands for end.gcode:

```
;End GCode
M104 S0 ;extruder heater off
M140 S0 ;heated bed heater off (if you have it)
G91 ;relative positioning
G1 E-1 F300 ;retract the filament a bit before lifting the nozzle, to release some of the pressure
G1 Z+0.5 E-5 X-20 Y-20 F{travel_speed} ;move Z up a bit and retract filament even more
G28 X0 Y0 ;move X/Y to min endstops, so the head is out of the way
M84 ;steppers off
G90 ;absolute positioning
```

Use the following commands for preSwitchExtruder.gcode:

```
;Switch between the current extruder and the next extruder, when printing with multiple extruders.
;This code is added before the T(n)
```

Use the following commands for postSwitchExtruder.gcode:

```
;Switch between the current extruder and the next extruder, when printing with multiple extruders.
;This code is added after the T(n)
```

Use the following commands for start2.gcode:

```
;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height} Walls: {wall_thickness} Fill: {fill_density}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament_weight}g
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to add your own bed temperature line
;M104 S{print_temperature} ;Uncomment to add your own temperature line
;M109 T1 S{print_temperature2} ;Uncomment to add your own temperature line
```

```
;M109 T0 S{print_temperature} ;Uncomment to add your own temperature line
G21 ;metric values
G90 ;absolute positioning
M107 ;start with the fan off
G28 X0 Y0 ;move X/Y to min endstops
G28 Z0 ;move Z to min endstops
G1 Z15.0 F{travel_speed} ;move the platform down 15mm
T1 ;Switch to the 2nd extruder
G92 E0 ;zero the extruded length
G1 F200 E10 ;extrude 10mm of feed stock
G92 E0 ;zero the extruded length again
G1 F200 E-{retraction_dual_amount}
T0 ;Switch to the first extruder
G92 E0 ;zero the extruded length
G1 F200 E10 ;extrude 10mm of feed stock
G92 E0 ;zero the extruded length again
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...
```

Use the following commands for end2.gcode:

```
;End GCode
M104 T0 S0 ;extruder heater off
M104 T1 S0 ;extruder heater off
M140 S0 ;heated bed heater off (if you have it)
G91 ;relative positioning
G1 E-1 F300 ;retract the filament a bit before lifting the nozzle, to release some of the pressure
G1 Z+0.5 E-5 X-20 Y-20 F{travel_speed} ;move Z up a bit and retract filament even more
G28 X0 Y0 ;move X/Y to min endstops, so the head is out of the way
M84 ;steppers off
G90 ;absolute positioning
```

Finally export the Gcode by doing "Save toolpath" and choose the folder of your SDCard on your computer.



Now insert the SDCard on the printer by using the jog button, navigate to "Print from SD" and choose the file. Your printer will start heating and will then print the object.