HBP USER MANUAL - V2

Table of Contents

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.

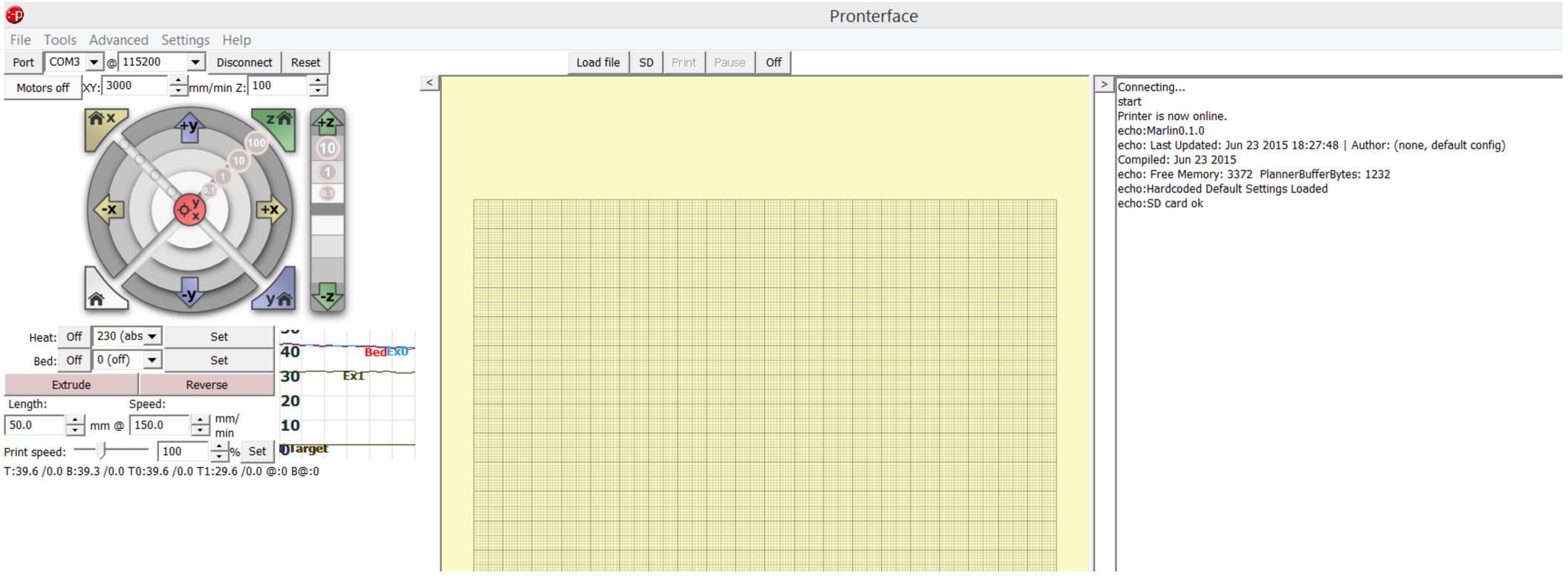
Software setup

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/

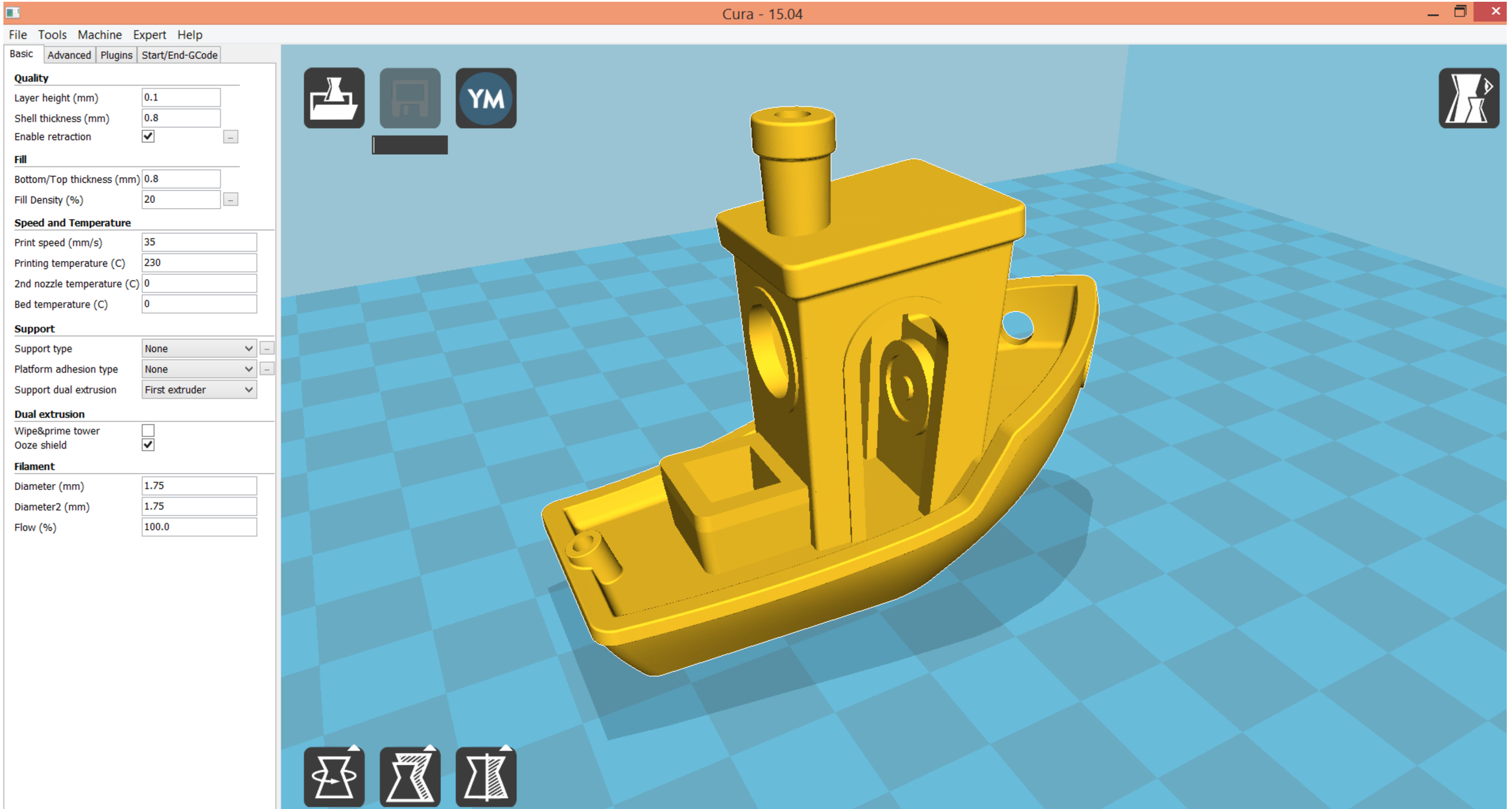
Pronterface screenshot:



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:


Cura - 15.04
File Tools Machine Expert Help
Basic Advanced Plugins Start/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) 230
2nd nozzle temperature (C) 0
Bed temperature (C) 0
Support
Support type None
Platform adhesion type None
Support dual extrusion First extruder
Dual extrusion
Wipe&prime tower
Ooze shield
Filament
Diameter (mm) 1.75
Diameter2 (mm) 1.75
Flow (%) 100.0
YM

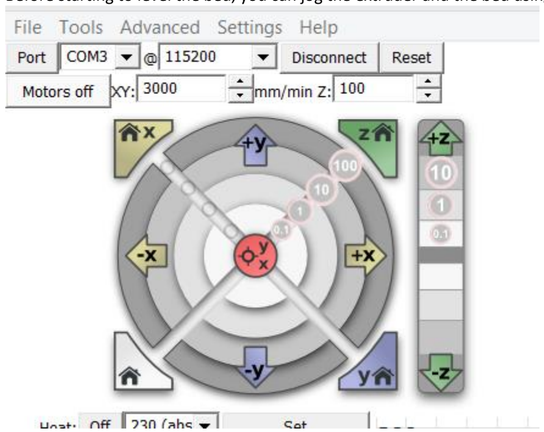
Preparing for first print

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

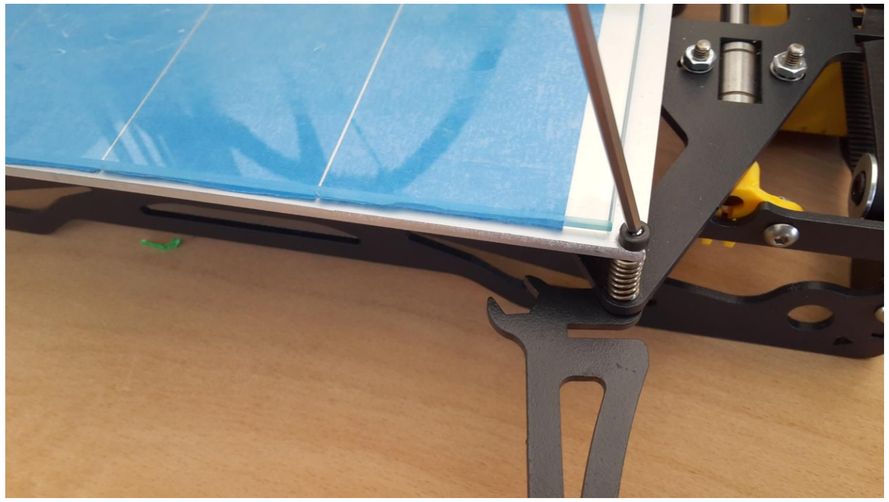
Bed calibration

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.

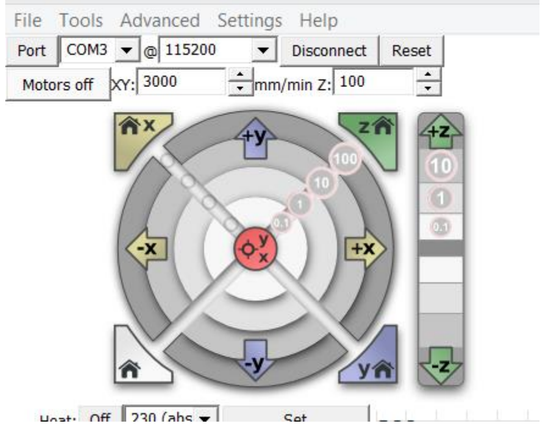


Now use the 3 screws to level the bed. You need to do it keeping the same distance from the noozle on all points of the bed.

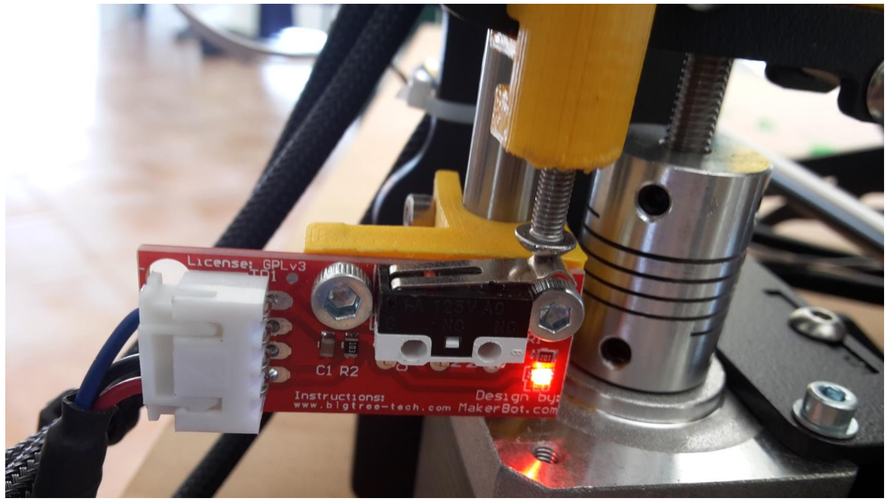


Next step is to adjust the home Z axis so the nozzle can stay at a distance of about 0.2mm from the bed. You can use a paper sheet as reference for the needed distance.

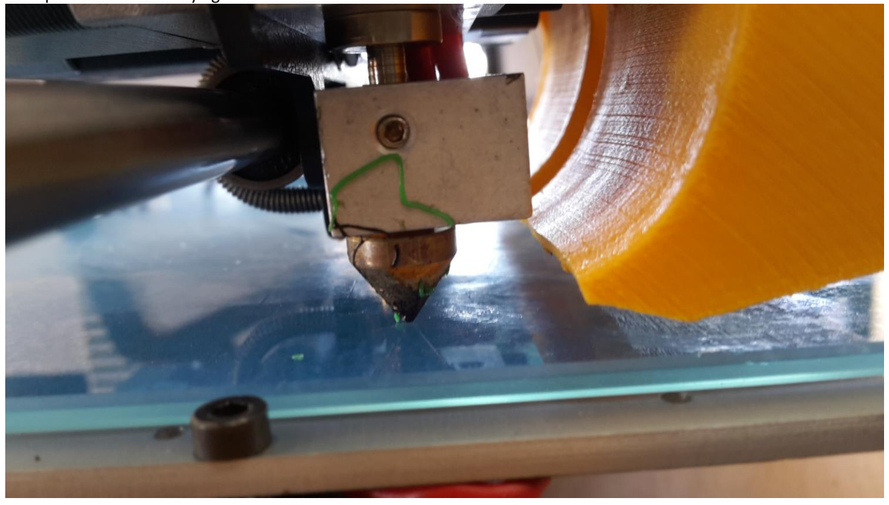
Use the Pronterface jog panel to home the Z axis:



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.

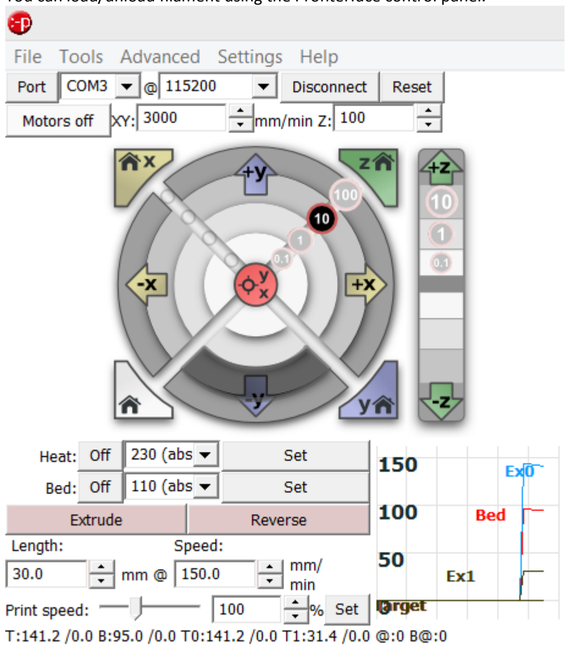


Example of the nozzle staying at about 0.2mm from the bed:



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.

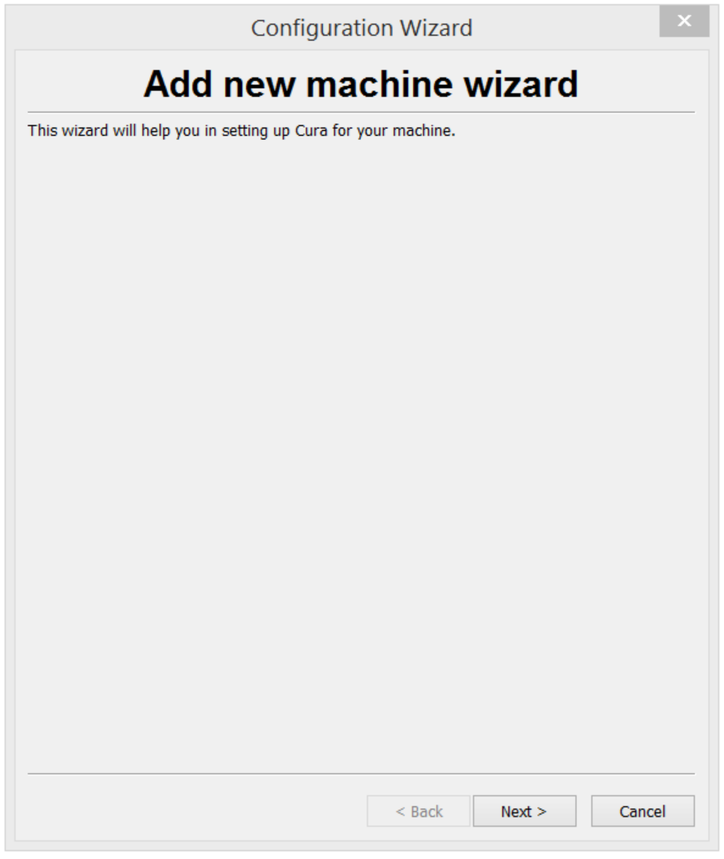
If you want to control the second extruder (ex 1), send the command "T1" on the Pronterface console in order to control the second extruder. Send the command "T0" to control again the first extruder.

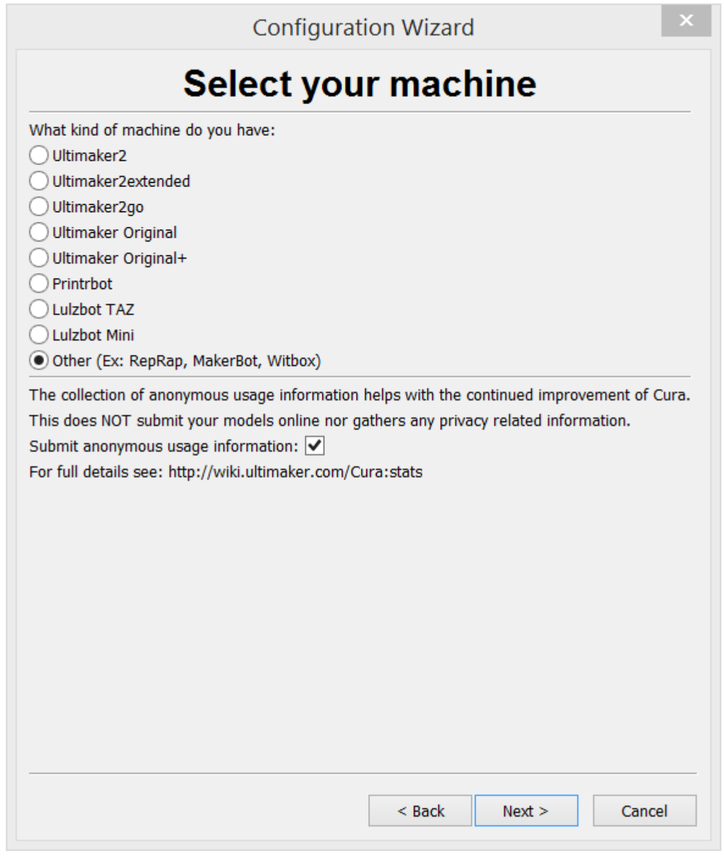
First print

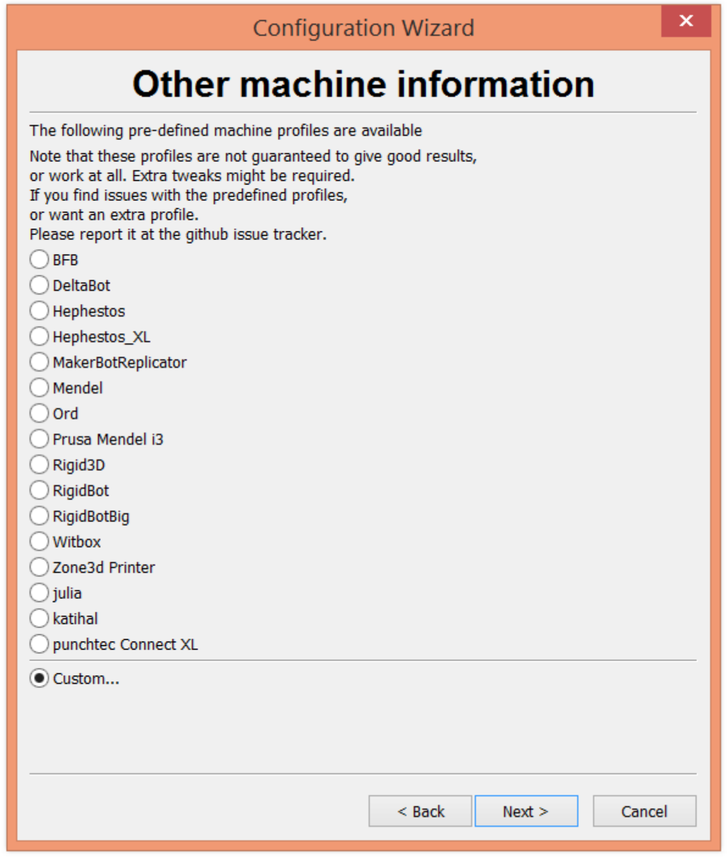
For our first print we will be using the free 3DBenchy 3D model - you can download it here: http://www.3dbenchy.com/
We will also configure Cura 15.04 to export the G-code. Finally we will save the G-code file on the SDCard and print it.

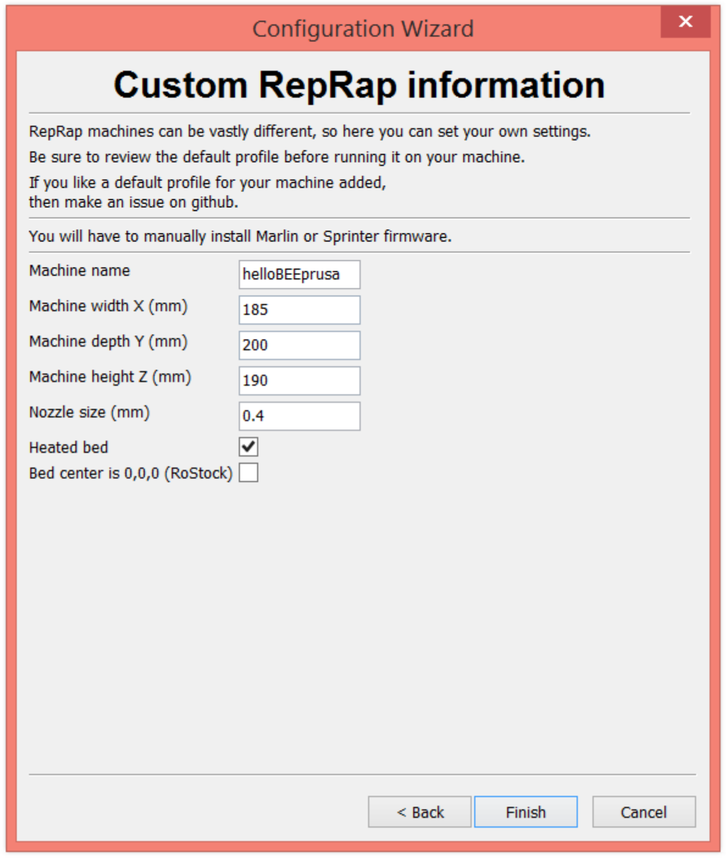
Add helloBEEprusa as a new machine on Cura 15.04

Go to "File --> Machine settings" and click on "Add new machine".
Follow the next screenshots and use the same options.

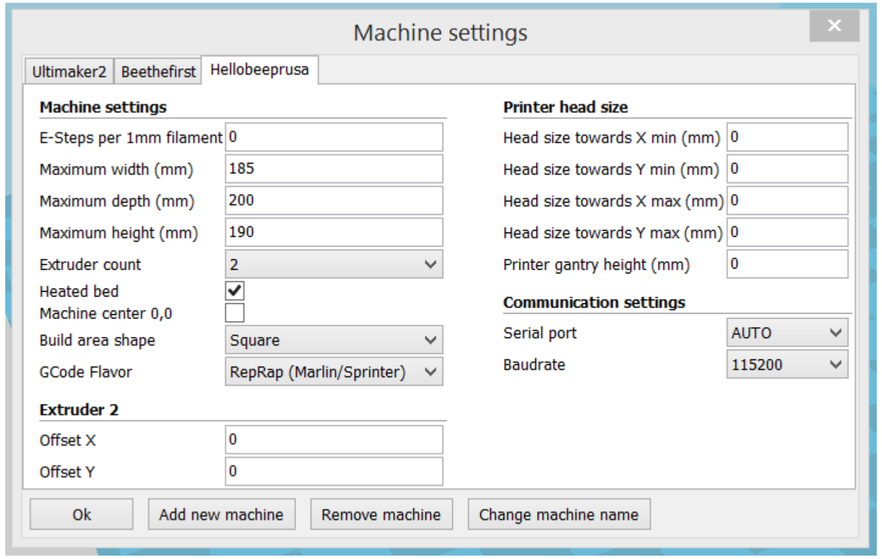

Configuration Wizard
Add new machine wizard
This wizard will help you in setting up Cura for your machine.
< Back
Next >
Cancel


Configuration Wizard
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
< Back
Next >
Cancel

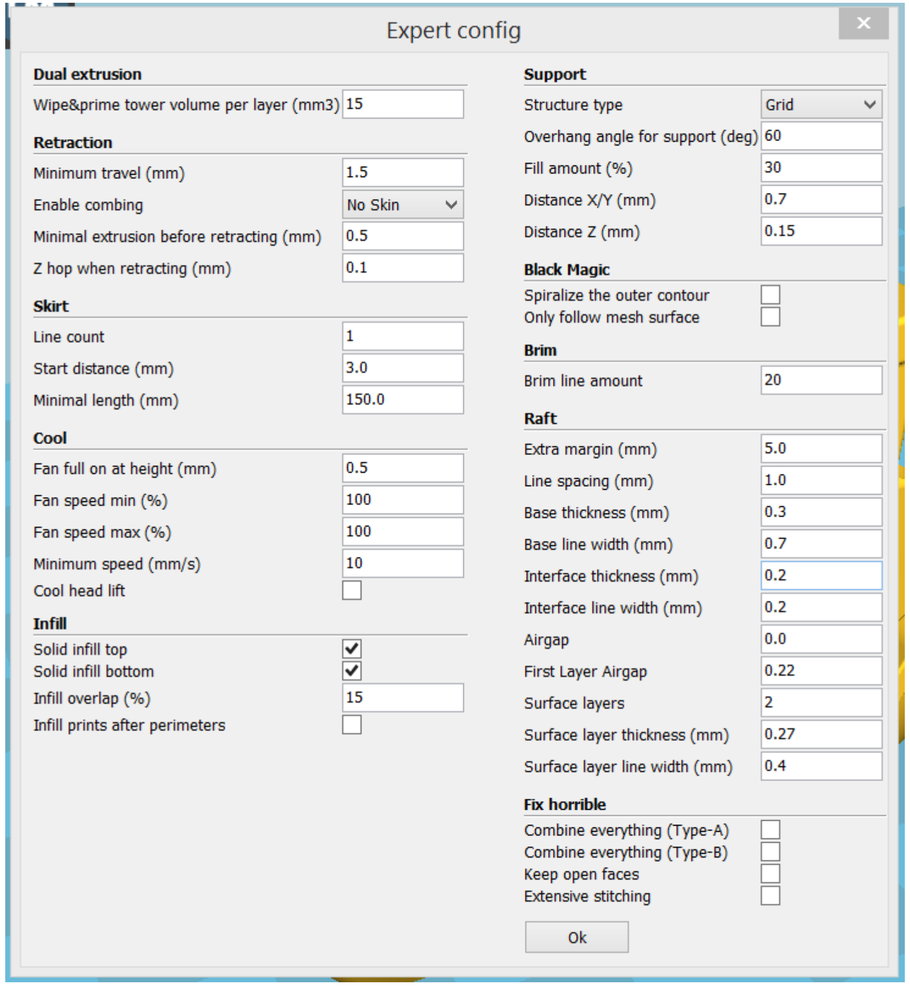

Configuration Wizard
Other machine information
The following pre-defined machine profiles are available
Note that these profiles are not guaranteed to give good results,
or work at all. Extra tweaks might be required.
If you find issues with the predefined profiles,
or want an extra profile.
Please report 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



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

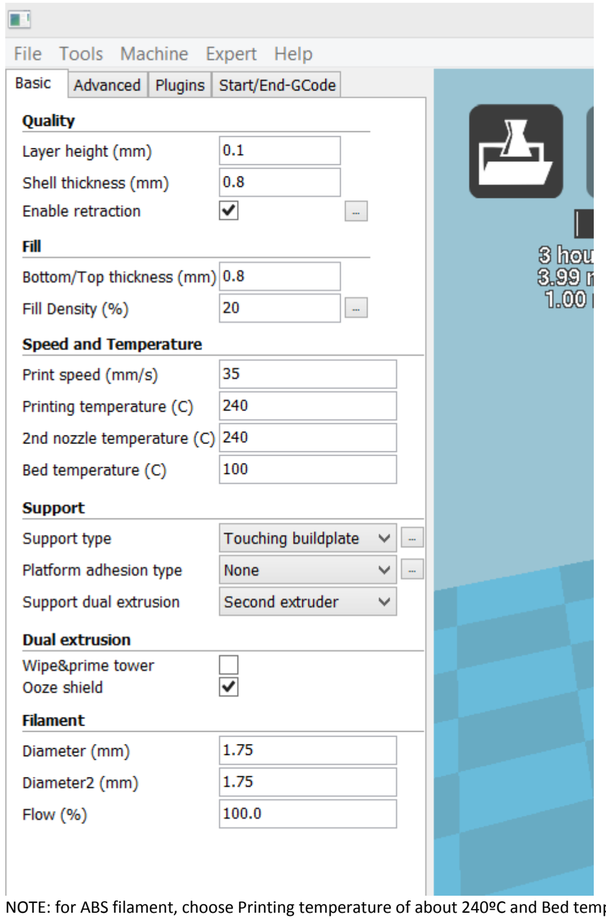


Access to "Expert --> Open expert settings..." and compare with the following screenshot:

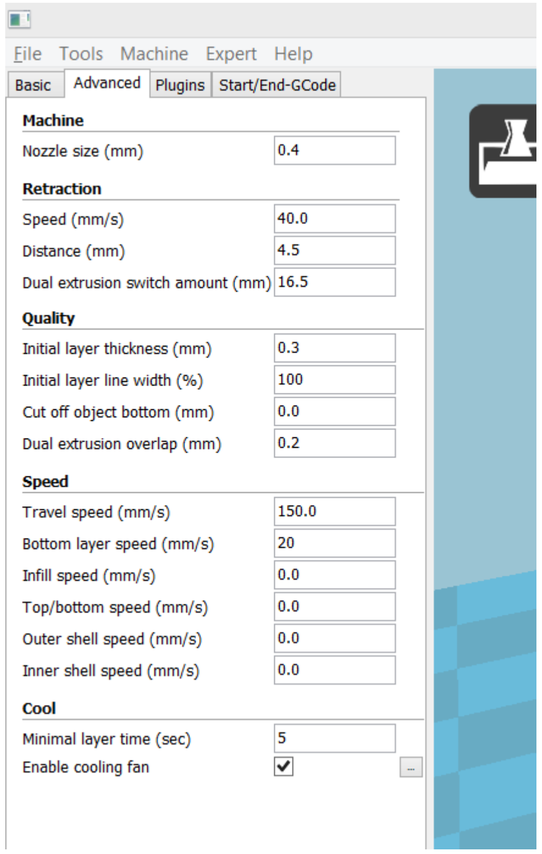


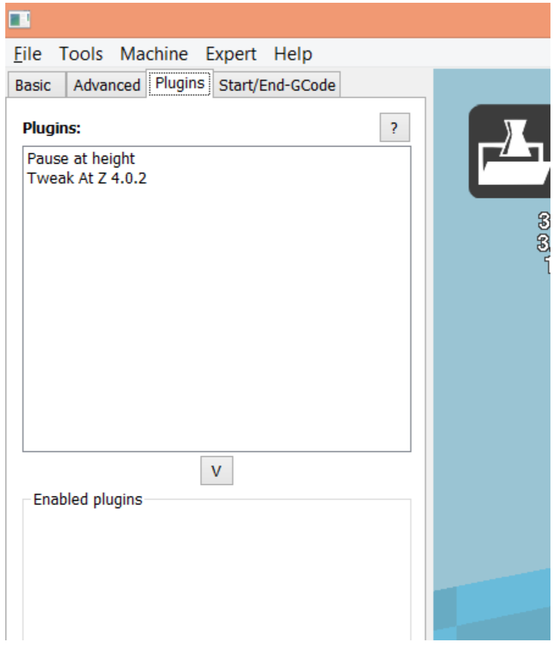
Configure settings and export GCode

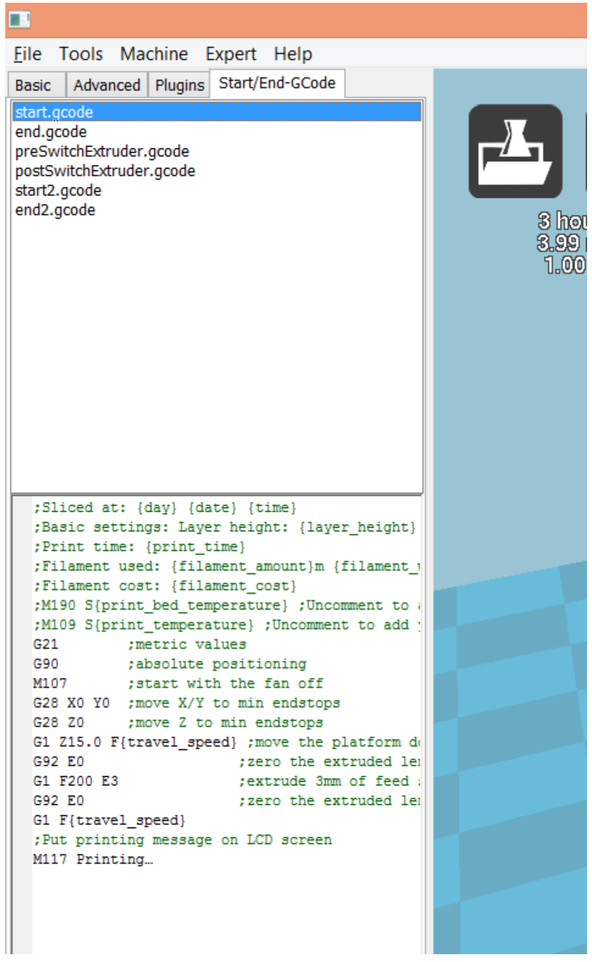
Follow these screenshots and use the same options.



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


File Tools Machine Expert Help
Basic Advanced Plugins Start/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


File Tools Machine Expert Help
Basic Advanced Plugins Start/End-GCode
Plugins:
?
Pause at height
Tweak At Z 4.0.2
V
Enabled plugins



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)

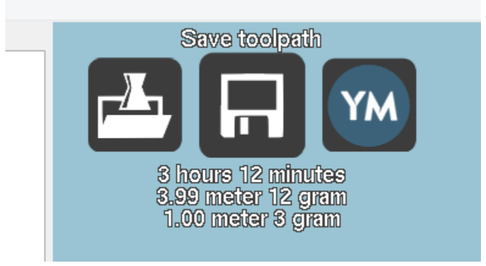
Use the following commands for postSwitchExtruder.gcode:
;Switch between the current extruder and the next extruder, when printing with multiple extruders.
;This code is added after the T(n)

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.