

USER MANUAL

TABLE OF CONTENTS

SOFTWARE SETUP

 Pronterface.....2

 Cura 15.04.....3

BED CALIBRATION

 Using Pronterface.....11

 Using LCD13

LOAD/UNLOAD FILAMENT

 Using LCD (extruder 0 only).....14

 Using PRONTERFACE.....15

FIRST PRINT

 Cura and SD Card.....16

 Pronterface and USB Cable.....16

GETTING STARTED WITH DUAL EXTRUSION

 How to setup the offset of the extruders.....17

 Set up the two extruders with the same height.....19

 Steps to print with two extruders.....21

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 operating systems.

SOFTWARE SETUP

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



PRONTERFACE

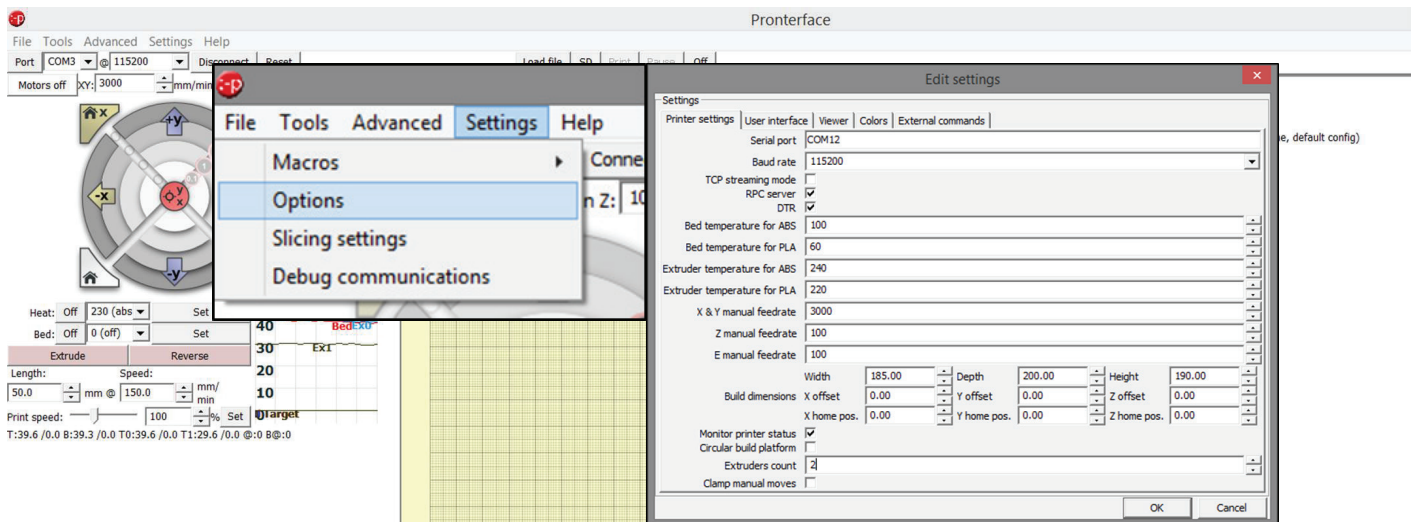
This software can be used to help you calibrate the print bed, load/unload filament and do other operations.

You can download it from this link - version "Printrun-Win-Slic3r-03Feb2015":

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

To make sure that the Pronterface is correctly configured, follow the next steps:

Go to "Settings" and select "Options", and make sure to put these settings in "Edit settings".

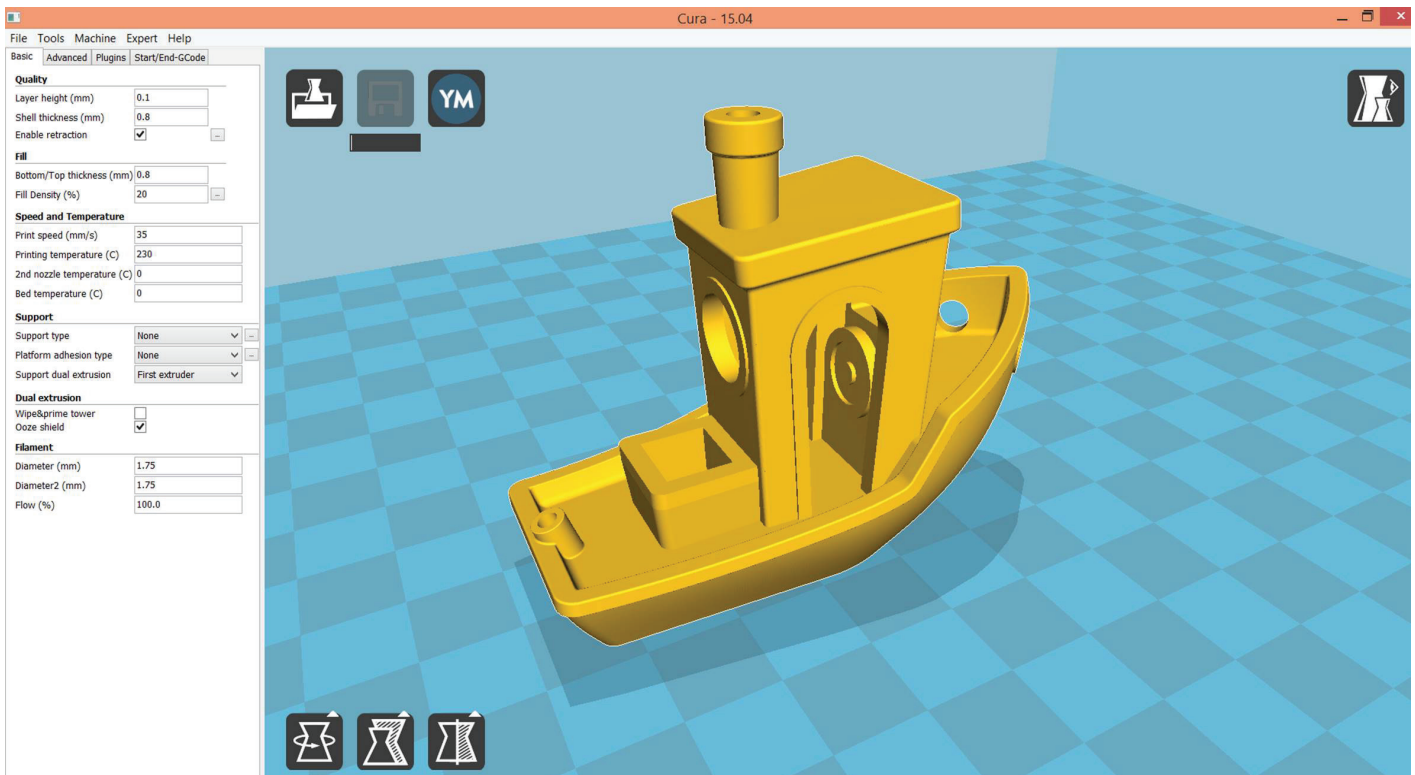




In order to generate G-code for printing, you can use Cura 15.04.

Please download only related versions from this link:

<https://ultimaker.com/en/products/cura-software/list>



Cura 15.04 screenshot

Before you start using this software you need to create a correct profile.
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.

The image displays four sequential screenshots of the Cura Configuration Wizard, illustrating the process of adding a new machine profile.

- Configuration Wizard - Add new machine wizard:** The first screen shows the title "Add new machine wizard" and a brief instruction: "This wizard will help you in setting up Cura for your machine." It includes navigation buttons: "< Back", "Next >", and "Cancel".
- Configuration Wizard - Select your machine:** The second screen, titled "Select your machine", asks "What kind of machine do you have:". It lists several options with radio buttons: Ultimaker2, Ultimaker2extended, Ultimaker2go, Ultimaker Original, Ultimaker Original+, Printbot, Lulzbot TAZ, Lulzbot Mini, and "Other (Eic RepRap, MakerBot, Witbox)". A note mentions that the collection of anonymous usage information helps with the improvement of Cura, with a checkbox for "Submit anonymous usage information:" which is checked. A link to the Ultimaker wiki is provided at the bottom.
- Configuration Wizard - Other machine information:** The third screen, titled "Other machine information", states that pre-defined machine profiles are available but not guaranteed to give good results. It asks the user to report issues on the GitHub issue tracker. A list of machine models is shown with radio buttons: BFB, DeltaBot, Hephestos, Hephestos_XL, MakerBotReplicator, Mendel, Ord, Prusa Mendel i3, Rigid3D, RigidBot, RigidBotBig, Witbox, Zone3d Printer, Julia, Kathal, and punchtec Connect XL. The "Custom..." option is selected.
- Configuration Wizard - Custom RepRap information:** The fourth screen, titled "Custom RepRap information", provides fields for machine-specific settings. It includes a text field for "Machine name" (filled with "helloBEEprusa"), and numeric input fields for "Machine width X (mm)" (185), "Machine depth Y (mm)" (200), "Machine height Z (mm)" (190), and "Nozzle size (mm)" (0.4). There is a checked checkbox for "Heated bed" and an unchecked checkbox for "Bed center is 0,0,0 (NoStock)".

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

The image shows the "Machine settings" dialog box in Cura, with the "Hellobeeprusa" profile selected. The dialog is divided into several sections:

- Machine settings:** Includes input fields for "E-Steps per 1mm filament" (0), "Maximum width (mm)" (185), "Maximum depth (mm)" (200), "Maximum height (mm)" (190), "Extruder count" (2), "Heated bed" (checked), "Machine center 0,0" (unchecked), "Build area shape" (Square), and "GCode Flavor" (RepRap (Marlin/Sprinter)).
- Printer head size:** Includes input fields for "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), and "Printer gantry height (mm)" (0).
- Communication settings:** Includes dropdown menus for "Serial port" (AUTO) and "Baudrate" (115200).
- Extruder 2:** Includes input fields for "Offset X" (0) and "Offset Y" (0).

At the bottom of the dialog are buttons for "Ok", "Add new machine", "Remove machine", and "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)15

Retraction

Minimum travel (mm)1.5

Enable combingNo Skin

Minimal extrusion before retracting (mm)0.5

Z hop when retracting (mm)0.1

Skirt

Line count1

Start distance (mm)3.0

Minimal length (mm)150.0

Cool

Fan full on at height (mm)0.5

Fan speed min (%)100

Fan speed max (%)100

Minimum speed (mm/s)10

Cool head lift

Infill

Solid infill top

Solid infill bottom

Infill overlap (%)15

Infill prints after perimeters

Support

Structure typeGrid

Overhang angle for support (deg)60

Fill amount (%)30

Distance X/Y (mm)0.7

Distance Z (mm)0.15

Black Magic

Spiralize the outer contour

Only follow mesh surface

Brim

Brim line amount20

Raft

Extra margin (mm)5.0

Line spacing (mm)1.0

Base thickness (mm)0.3

Base line width (mm)0.7

Interface thickness (mm)0.2

Interface line width (mm)0.2

Airgap0.0

First Layer Airgap0.22

Surface layers2

Surface layer thickness (mm)0.27

Surface layer line width (mm)0.4

Fix horrible

Combine everything (Type-A)

Combine everything (Type-B)

Keep open faces

Extensive stitching

Ok

CONFIGURE SETTINGS

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)
0.6

Fill Density (%)
10

Speed and Temperature

Print speed (mm/s)
30

Printing temperature (C)
220

2nd nozzle temperature (C)
220

Bed temperature (C)
60

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

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/End-GCode

Machine

Nozzle size (mm)
0.4

Retraction

Speed (mm/s)
40

Distance (mm)
4.5

Dual extrusion switch amount (mm)
10

Quality

Initial layer thickness (mm)
0.3

Initial layer line width (%)
100

Cut off object bottom (mm)
0.0

Dual extrusion overlap (mm)
0.15

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
☒

FileToolsMachineExpertHelp

BasicAdvancedPluginsStart/End-GCode

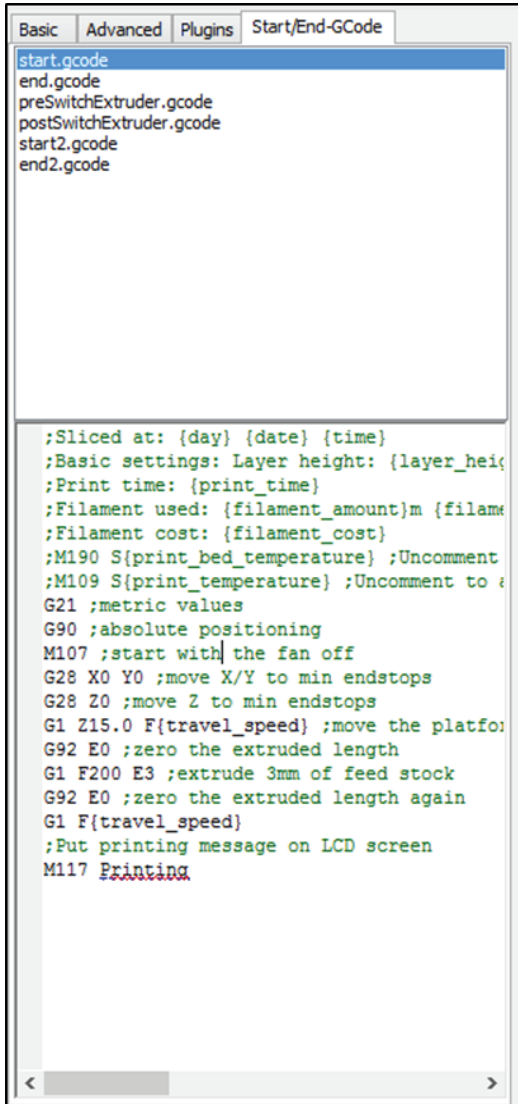
Plugins:

Pause at height
Tweak At Z 4.0.2

Enabled plugins

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 220°C and Bed temperature of about 60°C.

Follow these screenshots and copy the following commands.



```
start.gcode
end.gcode
preSwitchExtruder.gcode
postSwitchExtruder.gcode
start2.gcode
end2.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 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

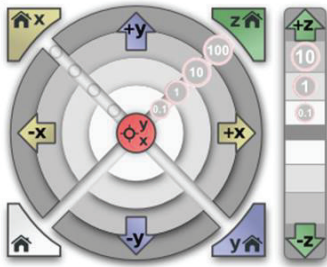
G90 ;absolute positioning

BED CALIBRATION

In order to be able to print, you first need to calibrate the bed.

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

For this you can use the Pronterface or the LCD.



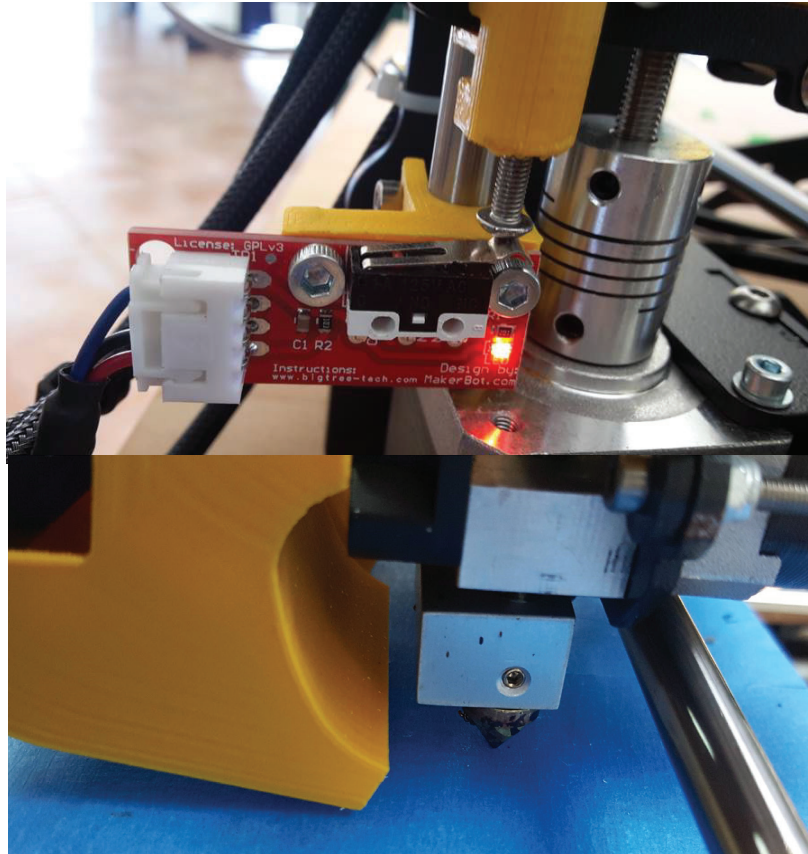
USING PRONTERFACE

Before starting to level the bed, you can jog the extruder and the bed using the jog buttons (picture below) 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.

Use the 3 screws to level the bed. This is done to give the same clearance from the nozzle on all points of the bed.



Next step is to adjust the home Z axis so the nozzle is 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 for this.



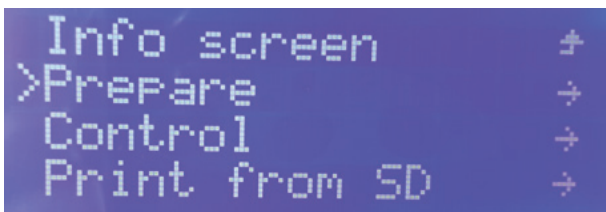
Example of the nozzle staying at about 0.2mm from the bed

USING THE LCD

You can use the LCD to level the bed also, follow the next steps:

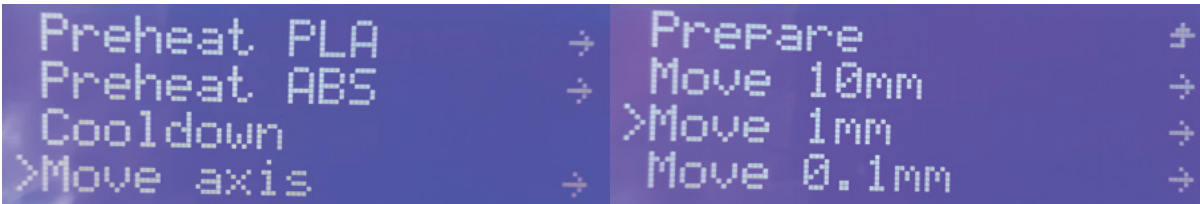
In the LCD push the button to have access to the "info screen";

In "info screen" select "Prepare";



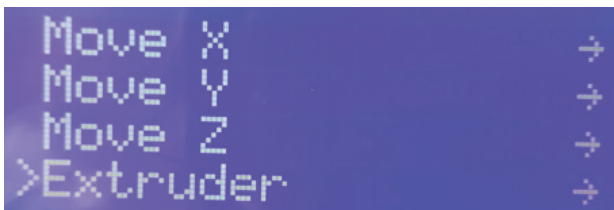
A screenshot of the LCD screen showing the 'Info screen' menu. The menu has four options: 'Info screen' (with a right arrow), '>Prepare' (with a right arrow), 'Control' (with a right arrow), and 'Print from SD' (with a right arrow).

In "Prepare" go to "Move axis" and select "Move 1mm";



A screenshot of the LCD screen showing the 'Prepare' menu. The menu has four options: 'Preheat PLA' (with a right arrow), 'Preheat ABS' (with a right arrow), 'Cooldown' (with a right arrow), and '>Move axis' (with a right arrow). The 'Move axis' option is highlighted, and a sub-menu is displayed to its right with four options: 'Prepare' (with a right arrow), 'Move 10mm' (with a right arrow), '>Move 1mm' (with a right arrow), and 'Move 0.1mm' (with a right arrow).

In "Move 1mm" select the axis you want to move;



A screenshot of the LCD screen showing the 'Move axis' sub-menu. The menu has four options: 'Move X' (with a right arrow), 'Move Y' (with a right arrow), 'Move Z' (with a right arrow), and '>Extruder' (with a right arrow).

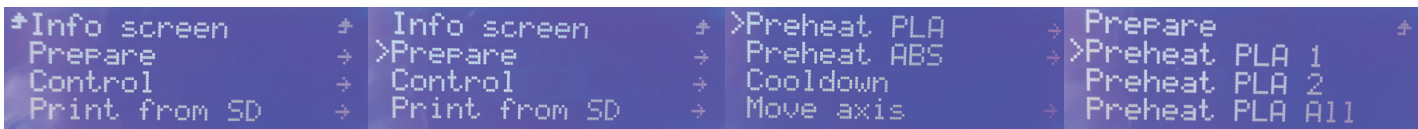
Rotate the button to the right or to the left to make the axis move, (note that the extruder motor only move if the extruder is heat but in this case you don't need to move it);

LOAD/UNLOAD THE FILAMENT

USING THE LCD (EXTRUDER 0 ONLY)

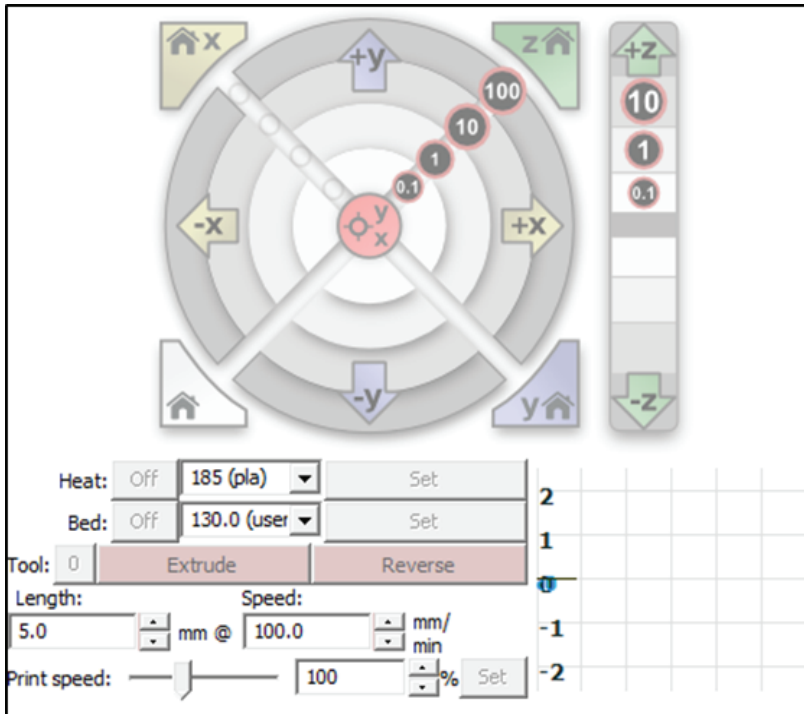
You can follow these steps to load/unload through the LCD, but this only works for the extruder 0:

1. In the LCD push the button to have access to the "info screen";
2. In "info screen" select "Prepare";
3. In "Prepare" go to "Preheat PLA" or "Preheat ABS";
4. In "Preheat PLA" select "Preheat PLA 1", the printing table and the extruder 0 will start heating;
5. After finishing heating, push the button to go to the "info screen" and select "Prepare";
6. In "Prepare" go to "Move axis" and select "Move 1mm";
7. In "Move 1mm" select "extruder";
8. In this screen you rotate the button to the right if you want to load and to the left if you want to unload.



USING PRONTERFACE

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

You can change the extruder by clicking on the button next to "Tool", Tool 0 is for the first extruder and Tool 1 is for the second 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/>

There are two ways to print:

CURA AND SDCARD

After we correctly configure Cura 15.04, we just need to import the STL file and export the Gcode.



We export the Gcode by doing "Save toolpath" and choose to save it in your SDCard.

Now insert the SDCard on the printer and with the LCD button, navigate to "Print from SD" and choose the file.

Your printer will start heating and will then print the object.

PRONTERFACE AND USB CABLE

First configure Pronterface and Cura, after that, use Cura to generate the GCode of the object you want to print.

Connect the printer through the USB.



Click on the "Connect" button and the printer will be connected to Pronterface, after that click on the "Load file" button and select the Gcode file that was generated by Cura before.

Now click on "Print", the printer will start to heat up and will print after that

Remember to never disconnect the USB cable or close the Pronterface, the printer will stop if you do that.

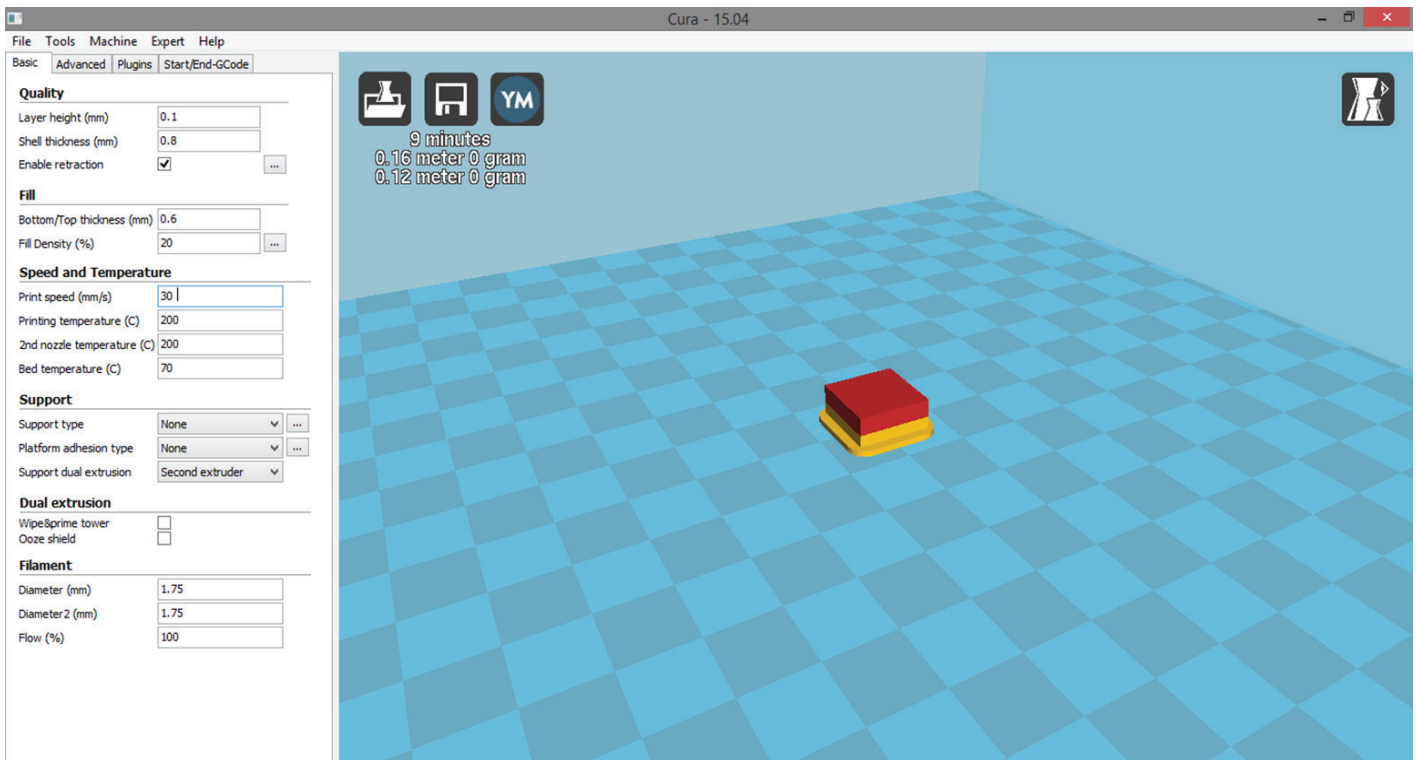
GETTING STARTED WITH DUAL EXTRUSION

Before you start printing with dual extruders, you need to setup the offset of the extruders.

HOW TO SET THE OFFSET OF THE EXTRUDERS

Download and drag it onto the printbed in Cura this file:

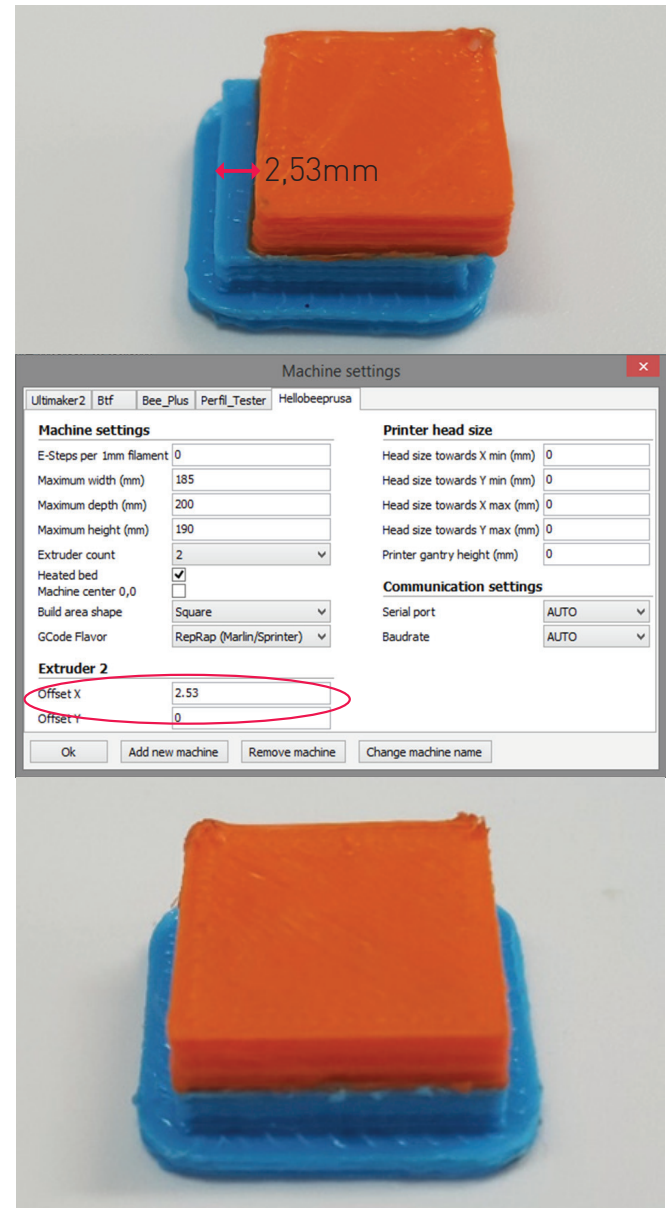
https://github.com/beeverycreative/helloBEEprusa-software/raw/master/dual_extruder_offset.amf



Export the G-Code and print. Grab a metric ruler and measure the distance of any axis offset that exists on the 3D-printed object. The Red rectangle and the Yellow rectangle should coincide. If they're not, measure the offset between them.

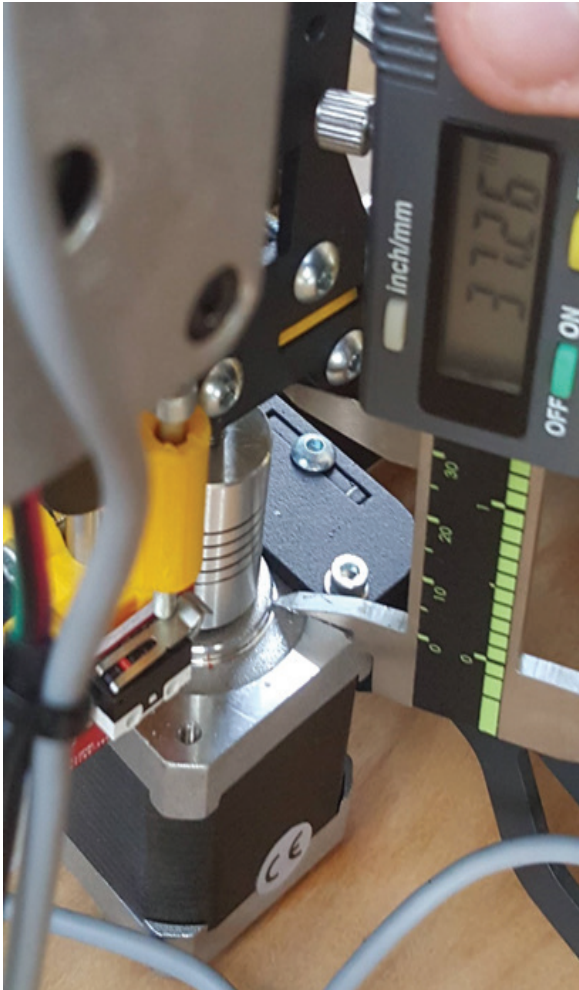
Now you need to setup in Cura the offset. Go to Machine → Machine Settings: under Extruder 2 change the Offset X and the Offset Y to the number that was measure before (in this example, offset of 2.53mm for X and 0 for Y).

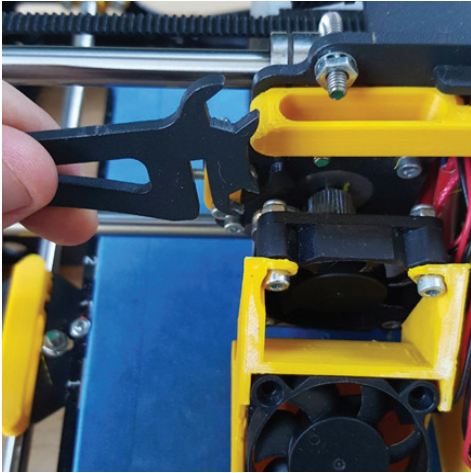
Test the offset and repeat the process if the print does not looks good. Example of printed part with a good offset setup:



SET UP THE TWO EXTRUDERS WITH THE SAME HEIGHT

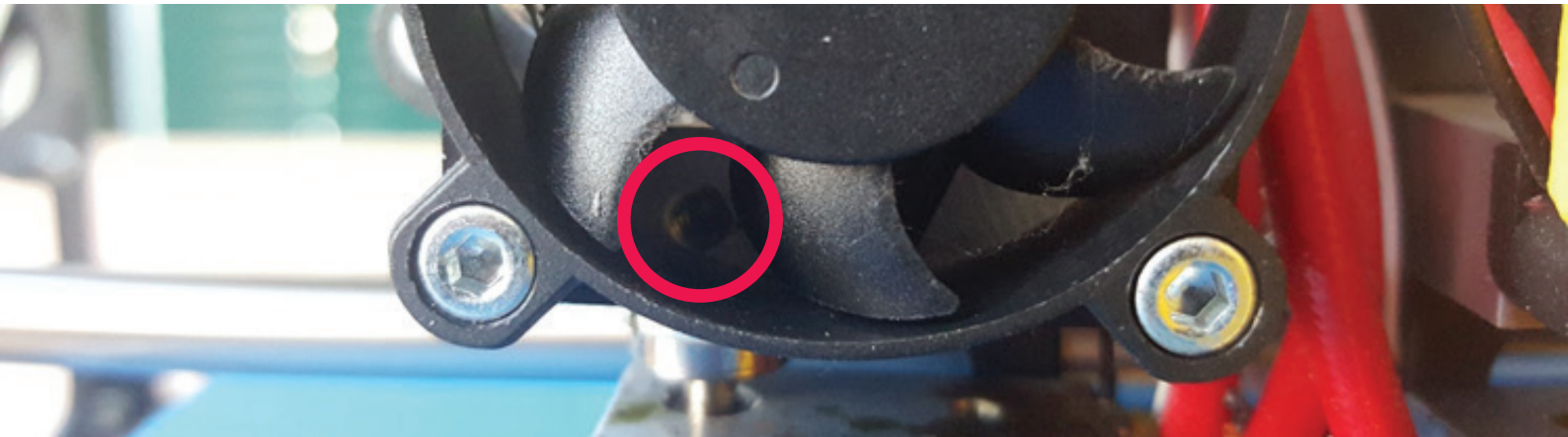
Ensure that the X carriage has the exact same height, using its Z motor as a reference, and ensure that the bed is calibrated with the lowest nozzle as reference.





With the extruders on the center of the heatbed and with the lowest nozzle touching the table, take off the blower off the extruder that isn't touching the table;

Unscrew the little screw and we can now lower the nozzle until it is touching the table too;



Screw it again and assemble the blower, now we have the nozzles at the same height.



STEPS TO PRINT WITH TWO EXTRUDERS

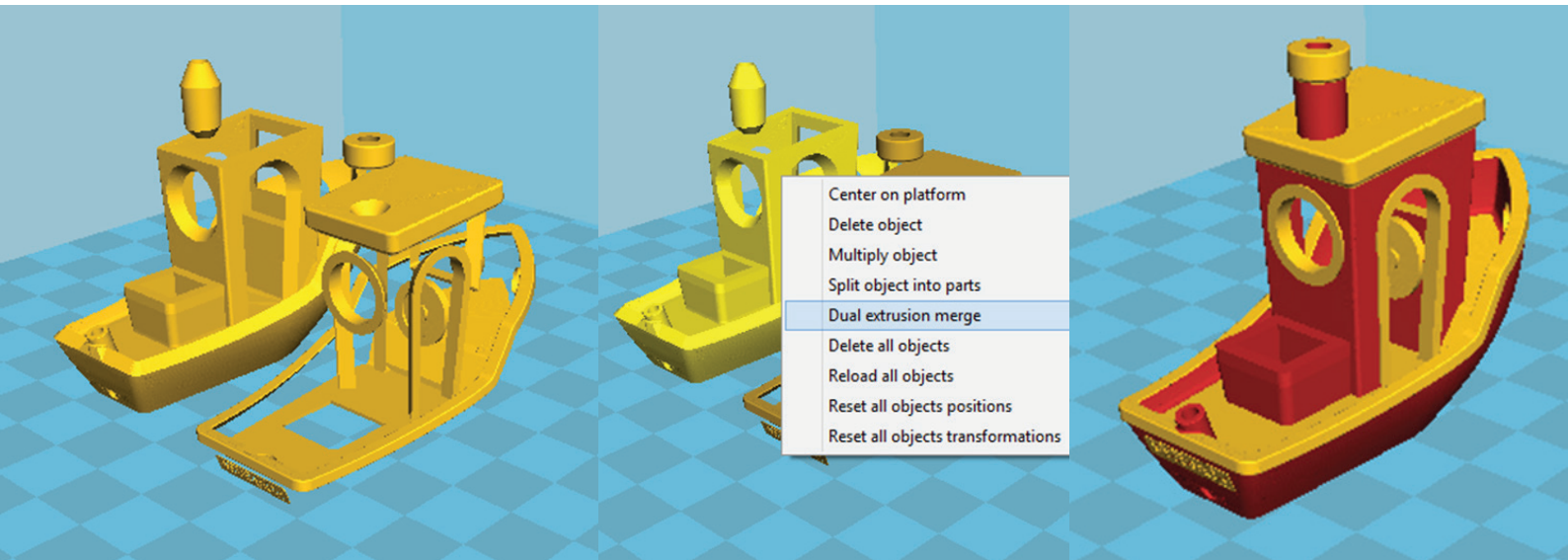
1. Load the two STL files into Cura, for instance these 3DBenchy files.

The first loaded STL file will be printed on extruder 0 and the second one on the extruder 1.

2. Select any object and with mouse right click and select "Dual extrusion merge":

3. After the merge, the model will have 2 different colours.

The yellow part will be printed by with extruder 0 and the red part by extruder 1:



4. Now export the ready-to-print G-Code.



www.facebook.com/beeverycreative



www.google.com/+beeverycreative



www.twitter.com/beeverycreative



www.linkedin.com/company/beeverycreative



www.instagram.com/beeverycreative



www.pinterest.com/beeverycreative

© 2016 BEEVERYCREATIVE
www.beeverycreative.com
All rights reserved.