



DIY 3D PRINTER KIT

CONTENTS

FIRST PRINT

Configuring Cura 15.04.6.	3
Extruder calibration	11
Bed calibration	13
Load/Unload filament	17
Using an SDcard	

GETTING STARTED WITH DUAL EXTRUSION

How to set extruder offset	19
How to set the two extruders at the same height	21
Steps to print with two extruders	23

OTHER OPTIONS

Uploading the firmware	24
Pronterface	27
Pronterface - Bed Calibration	28
Pronterface - First Print	28
Load/unload Filament (optional ways)	29

UPGRADE YOUR helloBEEprusa	

The information on this manual was written with software running on Microsoft Windows 10. However, the same software can be found for Mac OS and Linux, and the same instructions may work on either operating system.

V8_20170707



FIRST PRINT

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

There are a few ways to print but we strongly recommended using Cura and an SDcard.

CONFIGURING CURA 15.04.6

In order to generate the G-code for printing, you can use the Cura 15.04.6 Please download only related versions from this link: <u>https://ultimaker.com/en/products/cura-software/list</u>



Before you start using this software you need to create a correct profile. Add helloBEEprusa as a new machine on Cura 15.04.6.

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

Configuration Wizard	× Configuration Wizard	X Configuration Wizard	× Configuration Wizard
Add new machine wizard	Select your machine	Other machine information	Custom RepRap information
This wizerd will help you in setting up Cura for your machine.	West find of machine do you have: ● Ufmaker 2 # Ufmaker 2 Extended + Ufmaker 2 Extended Ufmaker 2 Extended Ufmaker 7 Grand Ufmaker 7 Grand The collection of anonymous usage information helps with the continued inprovement of Cura. This does NOT submit your models online nor gathers any privacy related information. Submit anonymous usage information: For full details see: http://wik.ultmaker.com/Cura:stats	The following per -defined machine profiles are evoluble instruction of the profiles are evoluble instruction of the profiles are evoluble if you for discuss with the predented profiles, or want an extra profile. Please report is after globulo issue trader. BPF Delabot Nephestos_NL Napido Nephestos_NL Napido Ord Prota Mendel I Plagidot_Zero Plagidot_Zero Plagidot_Zero Plagidot_Zero Plate Planter Plate Planter Plate Planter Plate	RepRag machines can be vasily different, so here you can set your own aettings. Be aze to releve the default profile for your machine. If you like a default profile for your machine added, them raide an aix on pithub. You will have to markally install Marlin or Sprinter firmware. Machine name IndeBEEEpruse Machine width X (mm) 185 Machine depth Y (mm) 200 Machine depth Y (mm) 190 Nozzle size (mm) 0.4 Heated bed Bed center is 0,0,0 (RoStock))

Access "Machine \rightarrow Machine settings..." and compare with the following screenshot.

ellobeeprusa			
Machine settings		Printer head size	
E-Steps per 1mm filament	0	Head size towards X min (mm)	0
Maximum width (mm)	185	Head size towards Y min (mm)	0
Maximum depth (mm)	200	Head size towards X max (mm)	0
Maximum height (mm)	190	Head size towards Y max (mm)	0
Extruder count	2 ~	Printer gantry height (mm)	0
Heated bed Machine center 0,0		Communication settings	
Build area shape	Square \checkmark	Serial port	AUTO ~
GCode Flavor	RepRap (Marlin/Sprinter) 🗸 🗸	Baudrate	250000 ~
Extruder 2			
Offset X	0.0		
Offset Y	0.0		

Go to "Expert \rightarrow Open expert settings..." and compare with the following screenshot.

Expert config

Retraction		Support
Minimum travel (mm)	1.5	Structure type
Enable combing	All 🗸	Overhang angle for support (deg
Minimal extrusion before retracting (mm)	0.5	Fill amount (%)
Z hop when retracting (mm)	0.0	Distance X/Y (mm)
Skirt		Distance Z (mm)
Line count	1	Black Magic
Start distance (mm)	3.0	Spiralize the outer contour
Minimal length (mm)	150.0	Unly follow mesh surface
Cool		Brim
Ean full on at height (mm)	0.5	Brim line amount
Ean road min (%)	100	Raft
Fan speed min (76)	100	Extra margin (mm)
Pari speeu max (%)	100	Line spacing (mm)
Minimum speed (mm/s) Cool head lift		Base thickness (mm)
Tofill		Base line width (mm)
Solid infill top		Interface thickness (mm)
Solid infill bottom		Interface line width (mm)
Infill overlap (%)	15	Airgap
Infill prints after perimeters		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

 \times

CONFIGURE SETTINGS

Follow these screenshots and use the same options.

File Tools Machine B	xpert Help	File Tools Machine Expert	Help	File Tools Machine Exp	pert Help
Basic Advanced Plugins	Start/End-GCode	Basic Advanced Plugins Start/	End-GCode	Basic Advanced Plugins	Start/End-GCode
Quality		Retraction		Plugins:	2
Layer height (mm)	0.2	Speed (mm/s)	15	Pause at height	
Shell thickness (mm)	1.2	Distance (mm)	1.5	Tweak At Z 4.0.2	
Enable retraction	····	Dual extrusion switch amount (mm)	4		
Fill		Quality			
Bottom/Top thickness (mm)	0.8	Initial layer thickness (mm)	0.3		
Fill Density (%)	10	Initial layer line width (%)	100		
Speed and Temperate	ıre	Cut off object bottom (mm)	0.0	Enabled pluging	V
Print speed (mm/s)	45	Dual extrusion overlap (mm)	0.15	Enabled plugins	
Printing temperature (C)	200	Speed			
2nd nozzle temperature (C)	200	Travel speed (mm/s)	150.0		
Support		Bottom layer speed (mm/s)	20		
Support type	None 🗸 …	Infill speed (mm/s)	0.0		
Platform adhesion type	None 🗸 …	Top/bottom speed (mm/s)	0.0		
Support dual extrusion	First extruder \sim	Outer shell speed (mm/s)	0.0		
Dual extrusion		Inner shell speed (mm/s)	0.0		
Wipe & prime tower		Cool			
Ooze shield		Minimal layer time (sec)	5		
Filament		Enable cooling fan	✓ ···		
Diameter (mm)	1.75				
Diameter2 (mm)	1./5				
How (%)	100.0				

Machine

NOTE: for ABS filament, recommended settings are: a printing temperature of about 240°C; bed temperature of 100°C or more.

For PLA filament, recommended settings are: a printing temperature of about 200°C; bed temperature of about 60°C.

6

Follow these screenshots and copy the following commands.



USE THE FOLLOWING COMMANDS FOR START.GCODE:

;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} M140 S{print_bed_temperature}; bed temperature line M109 S{print temperature} ;temperature line G21 :metric values G90 ;absolute positioning M107 :start with the fan off G28 ;move X/Y/Z to min endstops 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 M420 S1

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+0.3 E-5 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 G90 ;absolute positioning G1 Y180 M84 ;steppers off ;{profile_string}

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:

;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} M140 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 ;move X/Y/Z to min endstops 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+0.3 E-5 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 G90 ;absolute positioning G1 Y180 M84 ;steppers off ;{profile_string}

Before start printing, you just need to confirm that your machine is correctly configured and with the latest firmware.

EXTRUDER CALIBRATION

One very important step before you start printing is to ensure that the extruder is correctly aligned. In some cases, extra paint is present in the metal mainframe that causes a misalignment after assembling the extruder.

Because of this, you can have a extruder like the following example.



If this happens, just follow the next steps to correct this.

Use the four M3 plain washers, as referred to on page C7 of the Assembly Manual, to give a little distance between the extruder heat sink and the metal frame.

This is represented in the picture.



Use one plain washer on each of the cylinder-head screw M3X30. After you place the four washers, the extruder should be aligned.



BED CALIBRATION

Before your first print, you must calibrate the bed so the first layer of the print can adhere to the bed evenly, otherwise it may ruin your print.

For this you can use the LCD. Just follow the next steps:

On the LCD, push the button to access "info screen".

In "info screen" select "Prepare".



In "Prepare" go to "Move axis" and select the axis you want to move (X, Y or Z).



Rotate the button to the right or to the left to make the axis move.

Now, you use the 3 screws to level the bed. Remember that you need to keep the same clearance 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 sheet of paper as reference for the needed distance.



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

Make sure the clearance between the nozzle and the build plate is the same near the three calibration screws. If you find that the calibration is not satisfactory, repeat the process and check the clearance on all three points again.



SEMI-AUTOMATIC 9-POINT CALIBRATION

Info screen Print from SD Change filament >Level bed



This option in the LCD will help you to ensure that prints will adhere to the bed, even if the bed is not perfectly level. Before this, make sure the bed is calibrated. Now follow these steps to enable the "Level bed" option. In the "Info screen" select "Level bed".

Homing XYZ axis Allow movement to Finish and press twice to continue The machine will now home the axis, you need to press twice to advance to the first calibration point.

The X carriage will move to this point (point 0,0 - origin).

Move Z: +0.200

This image is what will appear at the LCD after you select "Level bed".

Move Z:



Rotate to adjust the height of the extruder and push it to select the value.

Note: You must make an adjustment at each point, even if a point doesn't need to be adjusted. Just press the button to accept

On each of the nine points, you must adjust the extruder height to make sure that the table and the extruder have a clearance of more or less 0.2mm.



You can see this points represented on the next picture.

Your printer should make a sound after finishing the leveling, if doesn't beep at the end of the calibration you need to manually save the settings by goint to "Control" select "Store memory.



Motion Filament >Store memory Load memory

LOAD/UNLOAD FILAMENT



- This option in the LCD will help you to load and unload filament in an easy way. For this you just need to follow the next steps.
- In the "Info screen" select "Change filament".

Now it is recommended that you select "Move to position" to ensure that the extruder goes to a safe position to change the filament.

>



The selected extruder will heat up;

When the extruder reaches the correct temperature, the printer will make a noise, you need to press and hold the button to continue, then the printer will perform the movement, Load or Unload and go back to the menu once finished. CHANGE FILAMENT Heating nozzle Please wait... Nozzle: 30/200

CHANGE FILAMENT

Press and hold to continue...



USING AN SDCARD

After we correctly configure Cura 15.04.6, we just need to import the STL file and export the Gcode.



We export the Gcode by doing "Save toolpath" and choosing to save it in your SDcard. Now insert the SDcard into 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.

GETTING STARTED WITH DUAL EXTRUSION

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

HOW TO SET EXTRUDER OFFSET

Download and drag the following file into Cura workspace: <u>https://github.com/beeverycreative/helloBEEprusa-software/raw/master/dual_extruder_offset.amf</u>



Export the G-Code and print. Grab a metric ruler and measure the distance of any axis offset in the 3D printed object. (E.g.: The red rectangle and the blue rectangle should coincide. If not, measure the offset between them as precisely as possible).

Now you need to setup the offset in Cura. Go to Machine → Machine Settings, under Extruder 2 change the Offset X and the Offset Y to the number that was measured 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 look good. On the right, an example of a printed object with a good offset setup.





HOW TO SET THE TWO EXTRUDERS AT THE SAME HEIGHT

Ensure that the X carriage has the exact same height, using its Z-axis 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 print bed and with the lowest nozzle touching the table, take off the printed part containing a fan. Remove the one pertaining to 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 in to the Cura, for instance these 3DBenchy files. The first loaded STL file will be printed on extruder 0 and the second one on extruder 1.
- 2. Select any object and with mouse right click, select the "Dual extrusion merge".
- After the merge, the model will have 2 different colours.
 The yellow part will be printed by extruder 0 and the red part by the extruder 1.



4. Now export the G-Code and it's ready to print.

OTHER OPTIONS

This section of the manual is to inform you of other possibilities of the helloBEEprusa.

UPLOADING THE FIRMWARE

Firmware is what controls the hardware and allows it to interact with outside devices.

These steps will help you to upload the firmware

1. Download the entire firmware folder, by clicking "Clone or download" then pressing "Download ZIP" from the following link: <u>https://github.com/beeverycreative/Marlin-BEEVERYCREATIVE</u>



Note: the page depicted here may change over time as it is updated frequently.

2. Download and install Arduino V1.6.8 (only this version) on your computer.

https://www.arduino.cc/en/Main/OldSoftwareReleases#1.0.x

Arduino 1.6.x, 1.5.x BETA

These packages are no longer supported by the development team.

1.8.1	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.8.0	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.13	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.12	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.11	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.10	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.9	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit Linux ARM	Source code on Github
1.6.8	Windows Windows Installer	MAC OS X	Linux 32 Bit Linux 64 Bit	Source code on Github

3. Unzip the firmware folder. In the Marlin folder, double-click on "Marlin.ino" to open the program.

📙 🖂 🚽 🛛 Marlin					- 🗆 ×
File Home Share View					~ (?)
Pin to Quick Copy Paste access	Move to v Organize	e Rename	New item •	Properties Open Open	Select all Select none Invert selection
$\leftarrow \rightarrow \times \land \square \rightarrow \text{This PC} \rightarrow \text{System (C)}$:) > Marlin			Search Marlin	0
Name All Million Control System (Control System)	Date modified 15/05/2017 16:43	Type CPP File	Size 12 KB	Search Wallin	^
🔥 macros	15/05/2017 16:43	H File	6 KB		
Makefile	15/05/2017 16:43	File	16 KB		
🔣 Marlin	15/05/2017 16:43	H File	14 KB		
💿 Marlin	15/05/2017 16:43	Arduino fi	le 2 KB		
🔥 Marlin_main	24/05/2017 16:50	CPP File	391 KB		
🔥 MarlinConfig	15/05/2017 16:43	H File	2 KB		
🔥 MarlinSerial	15/05/2017 16:43	CPP File	14 KB		
🔥 MarlinSerial	15/05/2017 16:43	H File	7 KB		
🔣 mesh_bed_leveling	15/05/2017 16:43	CPP File	2 KB		
🔣 mesh_bed_leveling	15/05/2017 16:43	H File	5 KB		
🔥 nozzle	15/05/2017 16:43	CPP File	8 KB		
🔥 nozzle	15/05/2017 16:43	H File	4 KB		
🔣 pins	15/05/2017 16:43	H File	15 KB		
oins_3DRAG	15/05/2017 16:43	H File	3 KB		
opins_5DPRINT	15/05/2017 16:43	H File	3 KB		
pins_AZTEEG_X1	15/05/2017 16:43	H File	1 KB		
pins_AZTEEG_X3	15/05/2017 16:43	H File	2 KB		
pins_AZTEEG_X3_PRO	15/05/2017 16:43	H File	4 KB		
pins_BAM_DICE_DUE	15/05/2017 16:43	H File	2 KB		
pins_BQ_ZUM_MEGA_3D	15/05/2017 16:43	H File	3 KB		
oins_BRAINWAVE	15/05/2017 16:43	H File	3 KB		
pins_BRAINWAVE_PRO	15/05/2017 16:43	H File	3 KB		
246 items 1 item selected 1,91 KB	15/05/0017 16.40	LI Cila	0 V D		

4. In menu tools select the board – Arduino Mega 2560. Please disconnect any USB equipment from your computer, with the exception of USB mouse and keyboard, and connect only your hello-BEEprusa with the USB cable.

5. Select the port;

6. Upload.



Arduino/Genuino Mega or Mega 2560, ATmega2560 (Mega 2560) on COM7

PRONTERFACE

This software can be used to help you calibrate the print bed, load/unload filament and carry out 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 these steps:

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



Pronterface screenshot

PRONTERFACE - BED CALIBRATION



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 as shown in the previous topic "Bed calibration using the LCD"p.13.



PRONTERFACE - FIRST PRINT

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

Connect the printer via USB.

Port	COM4 💌	@	250000	-	Connect	Reset
------	--------	---	--------	---	---------	-------

Load file	SD	Print	Pause	Off
-----------	----	-------	-------	-----

Click on the "Connect" button and the printer will be connect 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.

LOAD/UNLOAD FILAMENT (OPTIONAL WAYS)

You can follow these steps to load and unload using the LCD: In the LCD push the button to have access to the "info screen".

In "info screen" select "Prepare".



In "Prepare" go to "Preheat PLA" or "Preheat ABS".



In "Preheat PLA" select "Preheat PLA 0", the printing table and the extruder 0 will start heating or select "Preheat PLA 1", the printing table and the extruder 1 will start heating.



After finishing heating, push the button to go to the "info screen" and select "Prepare".



In "Prepare" go to "Move axis" and select "Extruder 0" or "Extruder 1.





In this screen you rotate the button to the right if you want to load and to the left if you want to unload.

UPGRADE YOUR helloBEEprusa

The information below will help you to understand what you need to do to update your helloBEEprusa.

First, check your serial number to know what applies to you.

1111300001->1111400124

UPGRADES:

- 2 x M5 brass nut;
- Newest printed parts;
- Upload the latest firmware;

1111500125-)1111500274

UPGRADES:

- Newest printed parts;
- Upload the latest firmware;

The STL for the latest printed parts are in the Forum: https://beeverycreative.com/forum/viewtopic.php?f=8&t=4

Just follow the last Assembly Manual to know how to apply the printed parts.

To upload the last firmware just follow the steps in p. 24.



www.facebook.com/beeverycreative

www.google.com/+beeverycreative

www.twitter.com/beeverycreative

www.linkedin.com/company/beeverycreative

www.instagram.com/beeverycreative

©2017 BEEVERYCREATIVE www.beeverycreative.com All rights reserved. Todos os direitos reservados