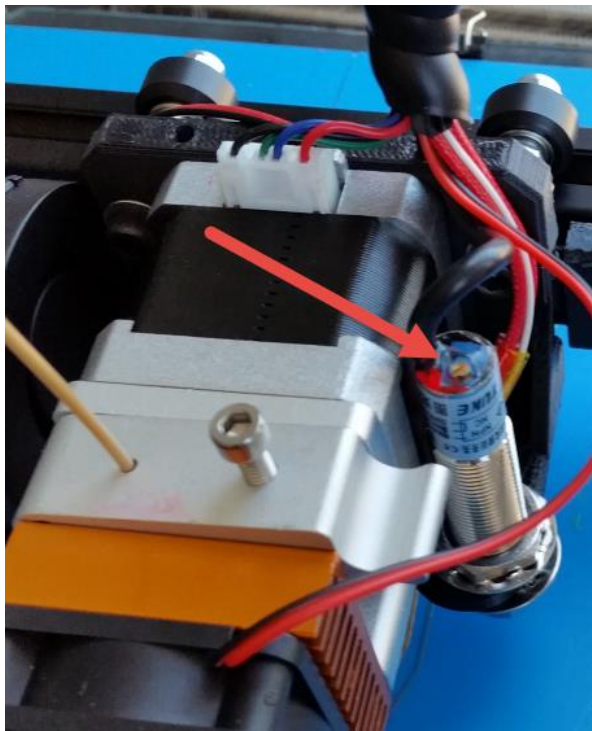


1. Check that the gap between the nozzle and the bed is good. To do that turn the printer on. On the LCD click the wheel and Navigate to "Prepare" and select "Auto home". It will bring the print head into the middle of the bed. Use standard printing paper to check the gap. The paper has to slide in you should feel slight friction. If the gap is ok please proceed to step 3 if not go to step 2.
2. If the gap need adjustment use the pot on top of the sensor as shown in the image below. If the gap is too big rotate the screw anticlockwise if the gap is too small rotate it clockwise. It is very sensitive and after each small adjustment repeat step #1.



3. Install the spool of filament on the top of printer. In the "Prepare" menu select "Preheat PLA" and once again click on the first choice in the menu.
4. Once it has pre-heated to 200C degrees (you can see it on the LCD left top corner) feed the filament through into the extruder and observing through the left side into the nozzle. The molten plastic will start coming out of the nozzle as you push the plastic through.
5. Download and install Cura 15.04 software to your computer.
https://ultimaker.com/en/products/cura-software/download-request/26/Cura_15.04.exe
6. After installing Cura please, start the application, go to "Machine" add new Machine and populate settings as in the image below.

Select your machine

What kind of machine do you have:

- ☐ Ultimaker2
- ☐ Ultimaker2extended
- ☐ Ultimaker2go
- ☐ Ultimaker Original
- ☐ Ultimaker Original+
- ☐ Printbot
- ☐ 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
- ☐ kathal
- ☐ punchtec Connect XL
- ☒ Custom...

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.

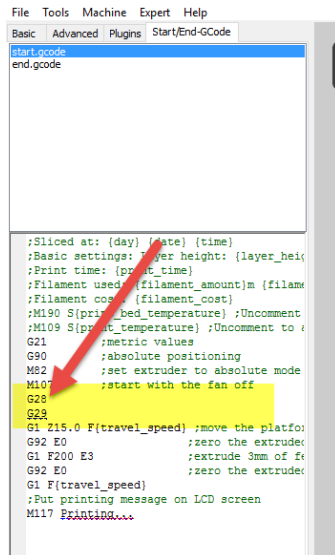
You will have to manually install Marlin or Sprinter firmware.

Machine name	<input type="text" value="RepRap"/>
Machine width X (mm)	<input type="text" value="250"/>
Machine depth Y (mm)	<input type="text" value="250"/>
Machine height Z (mm)	<input type="text" value="330"/>
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"/>

7. Download and import Wombot printer profile from here:

http://www.wombot.com.au/wp-content/uploads/2015/09/wombot_exilis_XL.zip

After import make sure that the Opening gcode has only G28 followed by G29 and nothing else.



```
File Tools Machine Expert Help
Basic Advanced Plugins Start/End-GCode
Start.gcode
end.gcode

;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament_weight}g
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to
;M109 S{print_temperature} ;Uncomment to
G21 ;metric values
G90 ;absolute positioning
M82 ;set extruder to absolute mode
M107 ;start with the fan off
G28
G29
G1 Z15.0 F{travel_speed} ;move the platform down
G92 E0 ;zero the extruder
G1 F200 E3 ;extrude 3mm of filament
G92 E0 ;zero the extruder
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...
```

The alternative to Cura can be Slic3r or Craftware. The links are available on www.wombot.com.au

8. Your software is ready for sliding your designs and preparing them for printing. To do that you have to open the .STL file that has your design. Save the .GCODE file to your SD card. Insert the SD card into the left side of the LCD screen and select from the menu “Print from SD” and select your file. The printer will start preparation for the print and will start printing once it has pre-heated.