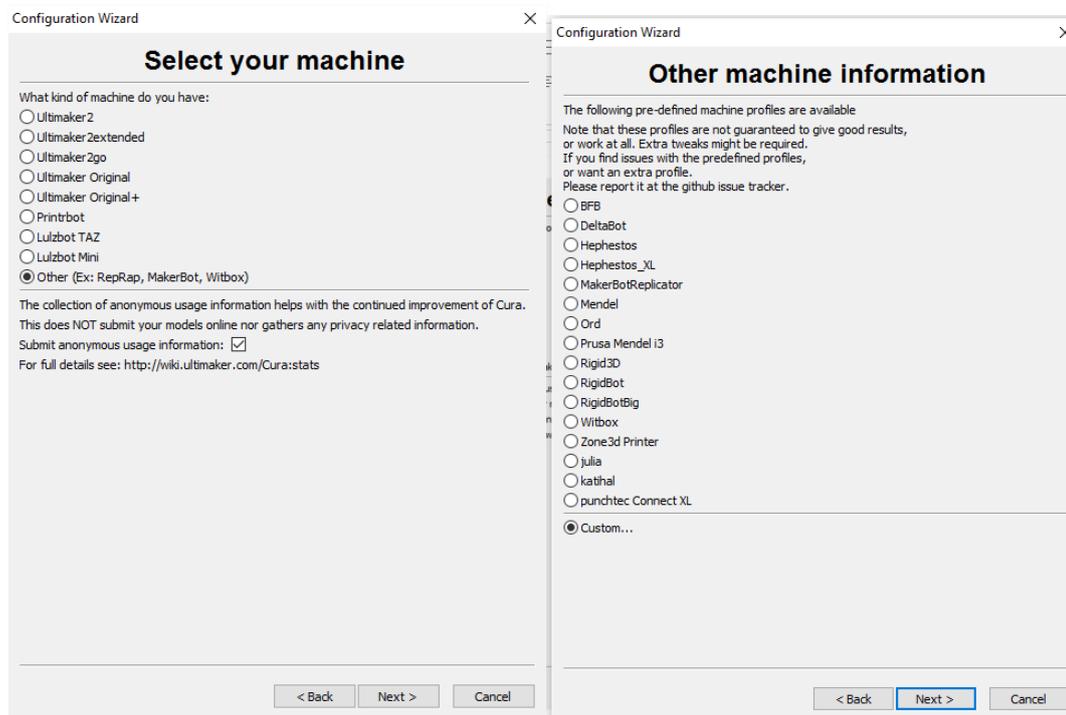
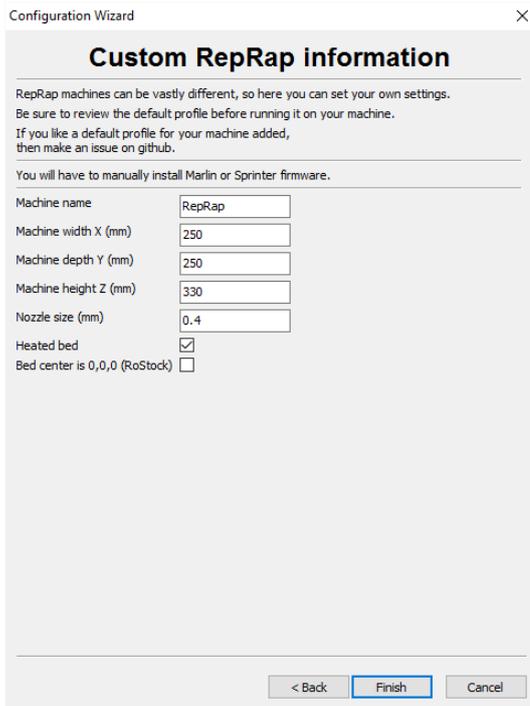


1. Watch the video on the SD card or <https://www.youtube.com/watch?v=YuxvNqK1Tv8>
2. Install the spool of filament on the top of printer. In the “Prepare” menu select “Preheat PLA” and once again click on the first choice in the menu.
3. Once it has pre-heated to 200C degrees (you can see it on the 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.
4. Download and install Cura 15.04 software to your computer.  
[https://ultimaker.com/en/products/cura-software/download-request/26/Cura\\_15.04.exe](https://ultimaker.com/en/products/cura-software/download-request/26/Cura_15.04.exe)
5. After installing Cura please, start the application, go to “Machine” add new Machine and populate settings as in the image below.

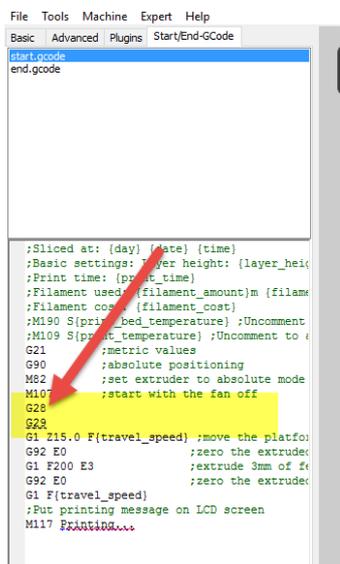




6. Download and import Wombot printer profile from here:

[http://www.wombot.com.au/wp-content/uploads/2015/09/wombot\\_exilis\\_XL.zip](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.



The alternative to Cura can be Slic3r or Craftware. The links are available on [www.wombot.com.au](http://www.wombot.com.au)

7. 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.

## Troubleshooting

### **Q. My BLTouch sensor is blinking**

A. The sensor starts to blink when it is in error mode. Most likely it was too low (too close to the bed) to perform self test. Cycle the power and try again

### **Q. My Z axis is going up and it tries to print in mid air. What is wrong?**

A. Check your BLTouch sensor. It is most likely in error mode – blinking. Cycle the power and try again

### **Q. I have got an error on the screen – "Error to far". What is wrong.**

A. The auto home was started from too high and it took too long for the Z axis to travel. The printer went into failsafe mode to prevent damage. Cycle the power and it should fix the issue.

### **Q. My printer starts printing off the bed – not where it is supposed to.**

A. It is high likelyhood that "Set offset" function was hit from the menu. Need to cycle the power to clear that error.

### **Q. Can I use other software package than Cura?**

A. Yes. It is open source printer and works with any slicing engine

### **Q. My nozzle goes into the bed. What do I do now?**

A. The offset is wrong. You have to correct it and try again.

### **Q. When the print starts it makes a mess – tangles on filament all over the place**

A. The nozzle too high. Make adjustment to the offset

### **Q. The print comes of the bed. What do I do**

A. Try not to print directly on glass. Use some sort of the surface treatment, it can be either Blue painters tape, Kapton tape, glue, Builtak or any other. It is not recommended to use hair spray as it would dust all the printer parts and it might affect the performance.

**Q. I want to enclose my printer. Can I do that.**

A. It is not recommended to do that, but if you want to you have to control temperature of the enclosure to be under 40C.