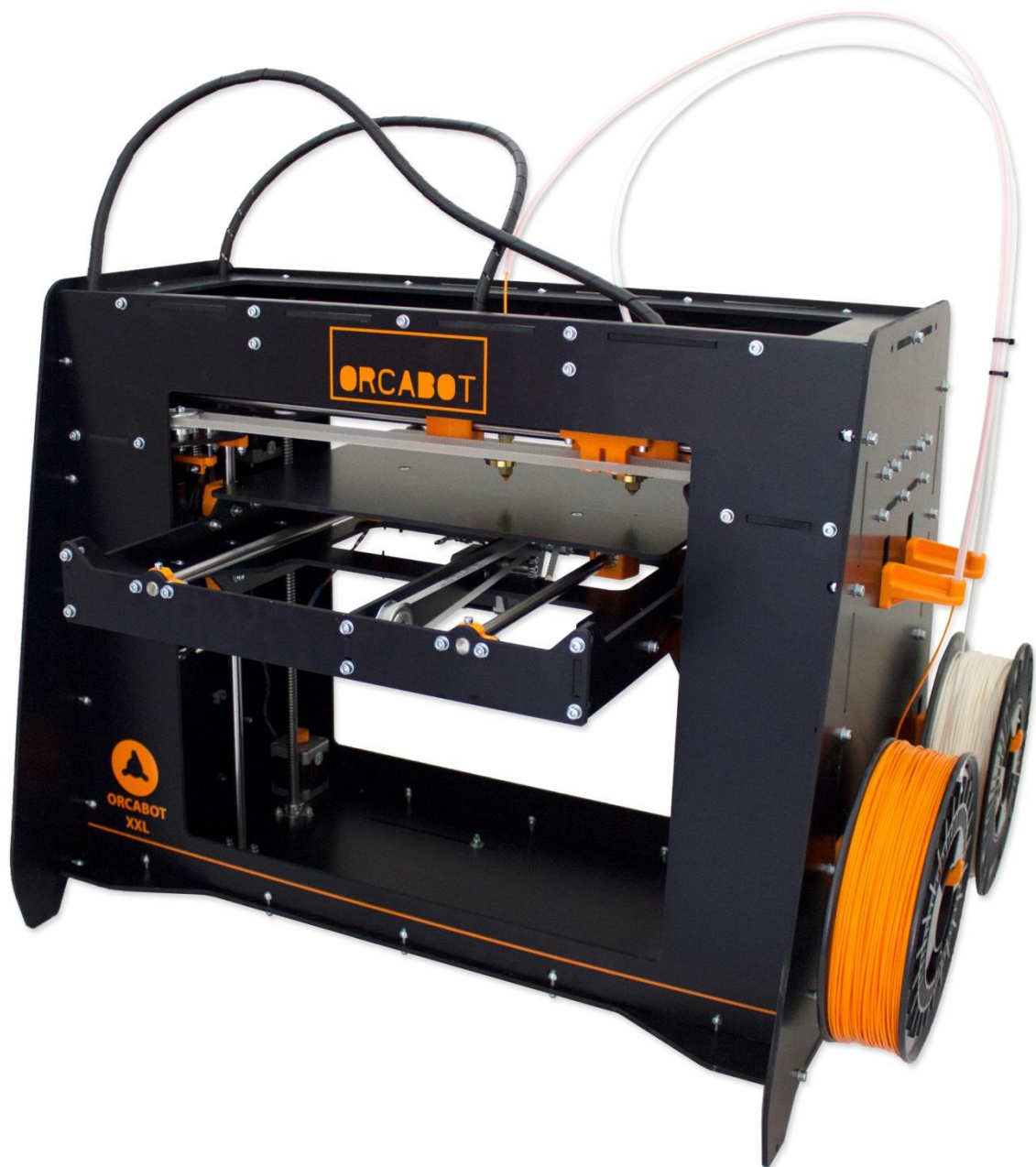


User manual: Orca XXL



User manual: Orca XXL

Version 1.1

May 2014





1. Content

1. Content.....	3
2. Welcome to the world of 3D printing	4
3. Safety.....	5
4. Declaration of conformity	Fout! Bladwijzer niet gedefinieerd.
5. Specifications.....	6
Printing	6
Dimensions.....	6
Temperature.....	6
Electrical	6
6. FFF 3D-print technology	7
7. Setting up the Orca.....	8
1. Installing the software.....	9
Step 1, driver install.....	9
Step 2, Repetier install	9
2. Printer Settings.....	12
3. Installing Print Profile	14
4. Prepare G-code for printing	18
5. Prepare G-code for printing with dual extruder	20
6. Printing	22
7. Replacing material.....	26



2. Welcome to the world of 3D printing

Thank you for choosing Orcabot as supplier for your 3D printing partner!

This user manual is designed to show you some basic steps in the 3D printing world. A world that is our future, full of challenges and opportunities.

It is essential that you read through this manual, as there are also some tips how to work safe and secure.

Let's start making amazing things!

3. Safety

	Safety alert symbols precede each safety message in this manual. These symbols indicate potential safety hazards that could harm you or others or cause product or property damage.	
	Warning:	The Orca generates high temperatures. Do not reach inside the machine when operating. Always allow the Orca to cool down first.
	Warning:	The Orca includes moving parts. Never reach the Orca when the machine is in operation.
	Warning:	Do not leave the Orca unattended during operation.
	Caution:	Only use the power supply provided with your Orca.
	Caution:	We recommend only to use materials supplied by Orcabot. Other materials can have unexpected properties with can influence the quality or worse.
	Caution:	In case of emergency disconnect power supply from wall socket.
	Caution:	Orca melts plastic during printing. Plastic odors are emitted during this operation. Make sure to set up the Orca in a well-ventilated area.

4. Specifications

Printing

Print technology:	Fused Filament Fabrication (FFF)
Layer resolution:	50 µm
Position resolution:	50 µm
Nozzle diameter:	0,35mm
Filament:	1,75mm Orcabot filament

Dimensions

Build volume:	360x280x230mm
Frame dimensions:	620x400x520mm
Total weight:	

Temperature

Ambient operating temperature:	10°C - 40 °C
Storage temperature:	0°C - 50 °C

Electrical

AC input:	88-264 VAC
Frequency range:	47-63 Hz
DC Voltage:	24V
Rated Current:	13A



5. FFF 3D-print technology

Fused filament fabrication (FFF) is an additive manufacturing technology commonly used for modeling, prototyping, and production applications.

FFF works on an "additive" principle by laying down material in layers; a plastic filament or metal wire is unwound from a coil and supplies material to produce a part.

The technology was developed by S. Scott Crump in the late 1980s and was commercialized in 1990.



6. Setting up the Orca

After unpacking the Orca, the next step is setting up the printer. A few things are important:

- The Orca must be placed on a smooth and level surface
- There must be an electric socket, recommended is a socket with an on/off switch
- It is recommended to choose a room or place which has a decent ventilation

The Orca package contains the Orca printer, a USB cable and a cable for the power supply. The USB cable is for connecting the printer to a computer.

Before testing and connecting the printer to the computer, choose a solid table or desk, to put the printer on. Switch the machine on and connect the Orca with a computer via the USB cable.

The computer will detect a new USB device on one of the COM ports.

7. Installing the software

Windows only, there is also an Apple OS package.

On the Orcabot website you can download the driver and software to control your Orca 3D printer.

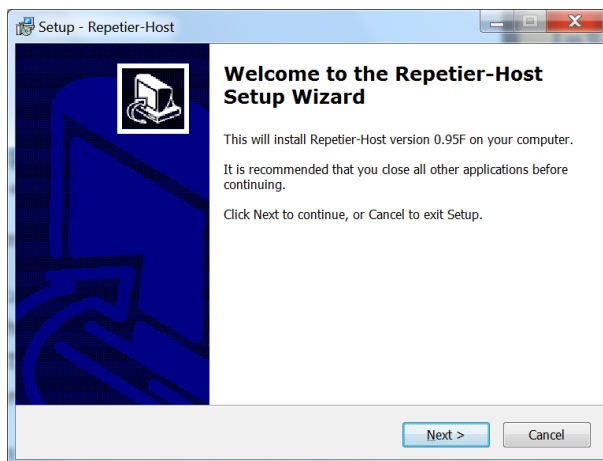
Step 1, driver install

Connect the printer to the computer. The computer will come up with a message “new hardware detected”. Follow the instructions in the wizard to install the driver: Marlin_CDC_driver_x86_x64.inf.

Step 2, Repetier install

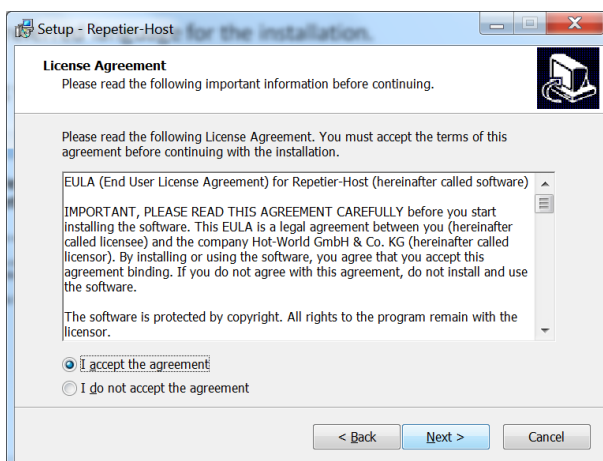
Open the folder “2-REPETIER-INSTALL”, select the “setupRepetierHost_0_95” and install it.

First, choose your preferred language for the installation.

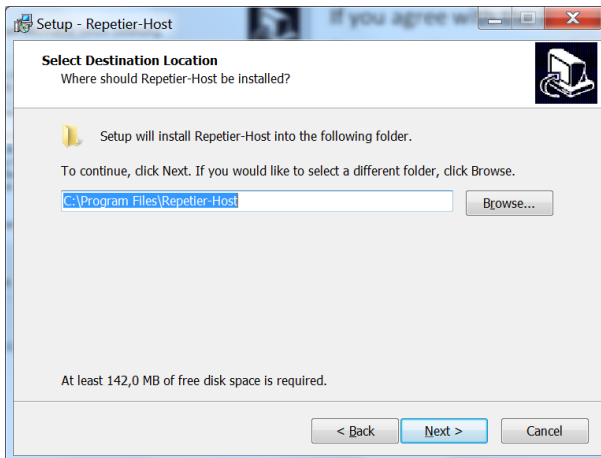


The following dialog will appear:

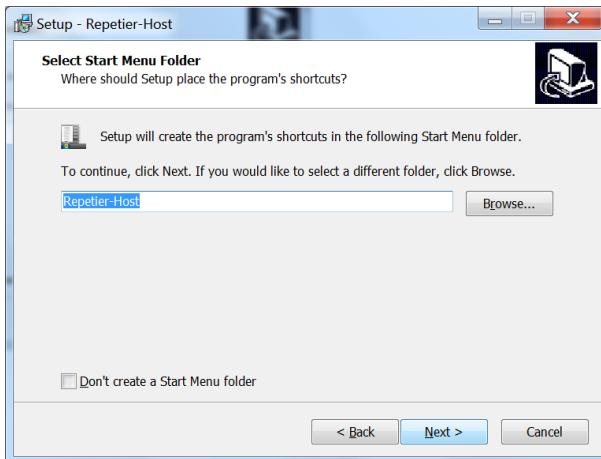
Click: “Next”



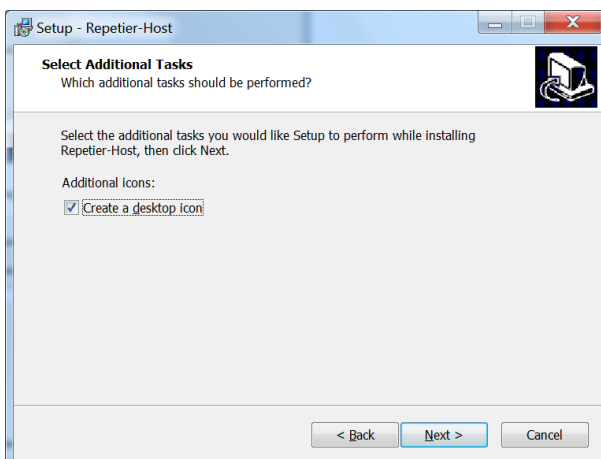
If you agree with the license agreement, select “I accept the agreement” and click “Next”.



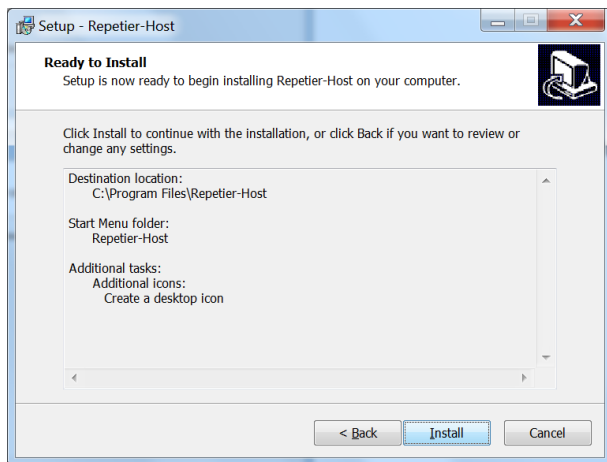
Select the destination folder and click “Next”.



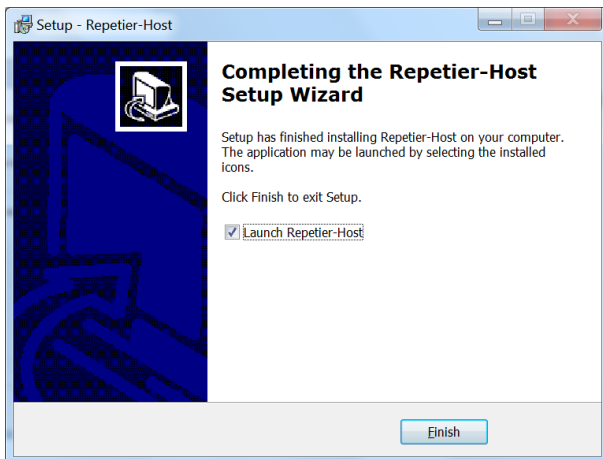
Select the start menu folder and click “Next”.



Select the checkbox to create a desktop icon and click “Next”.



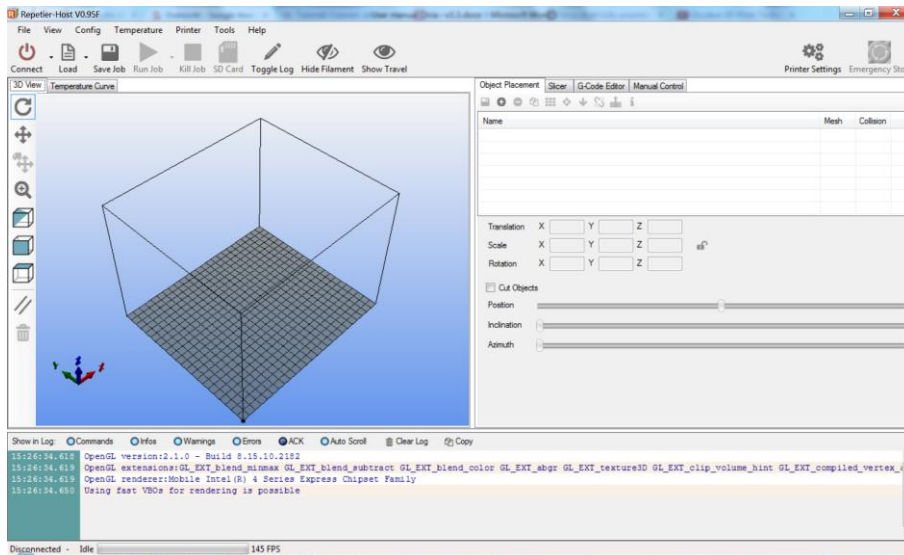
Click “Install” to proceed with the installation.



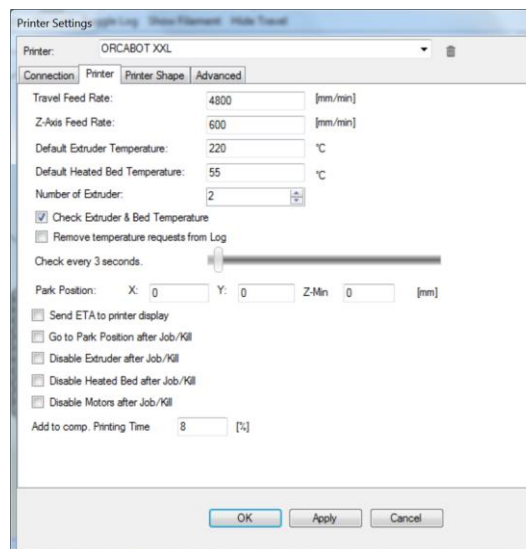
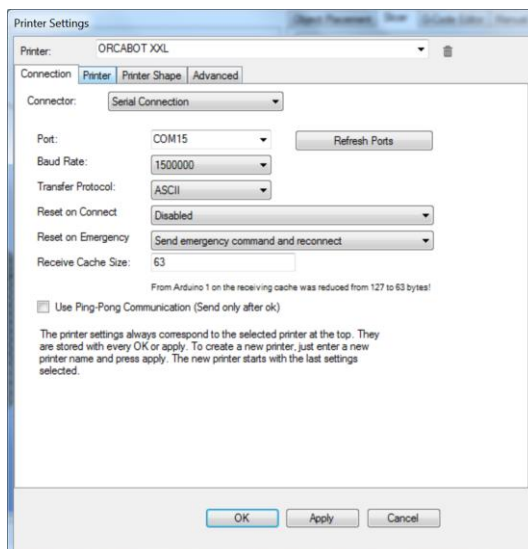
Finally click “Finish” to finish the installation.

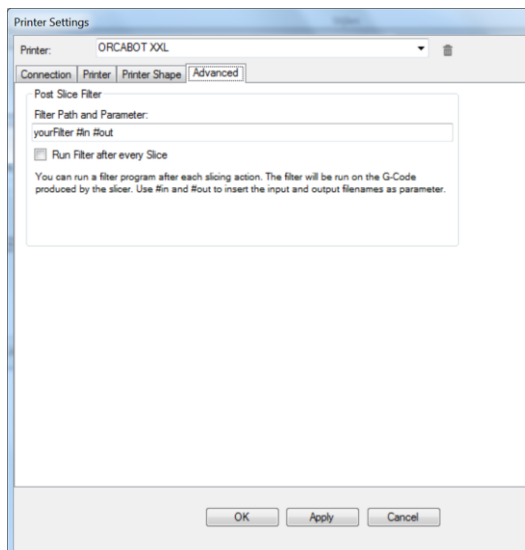
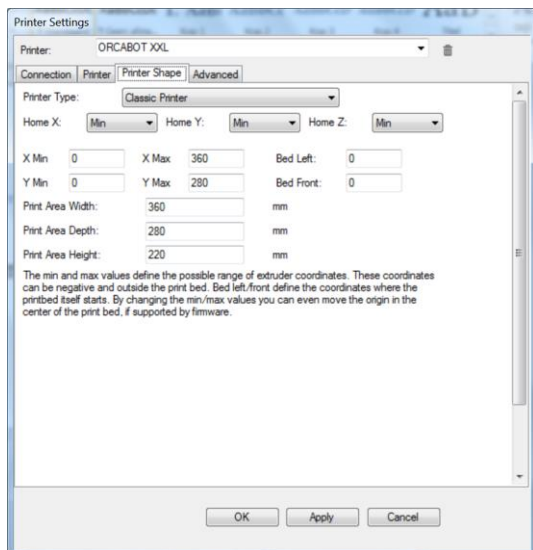
8. Printer Settings

Start Repetier Host and select “Printer settings” in the right top corner.



Please enter the following settings in the four TAB's and after that press “Apply” and “OK”.



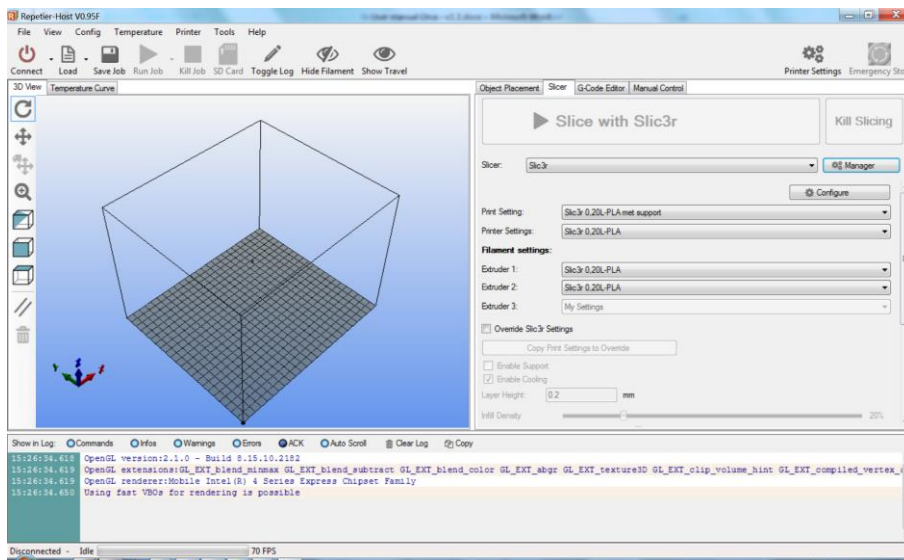


9. Installing Print Profile

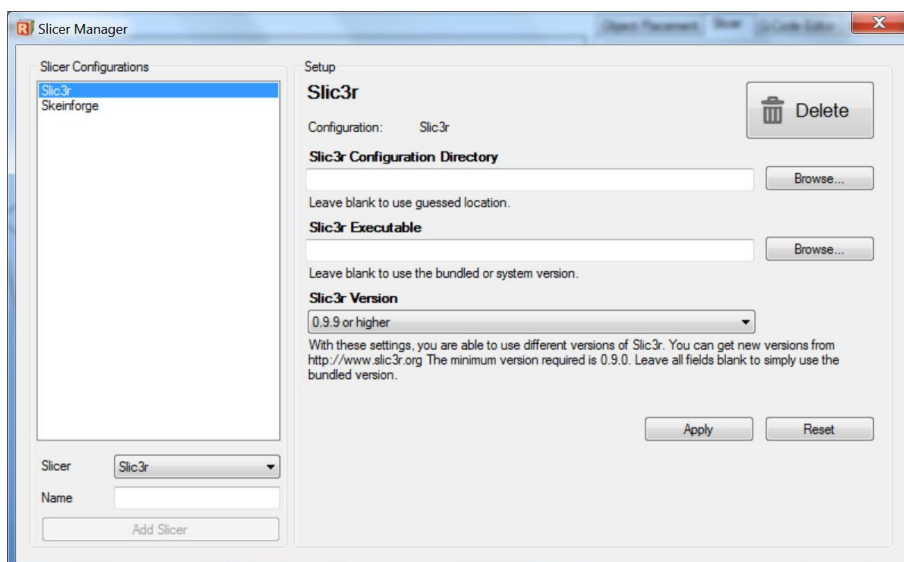
Next step is to install the Print Profiles in Repetier Host. These files include the layer height and settings for printing a specific material like: PLA, ABS, Nylon etc..

You can download the Repetier Print Profiles from the Orcabot website.

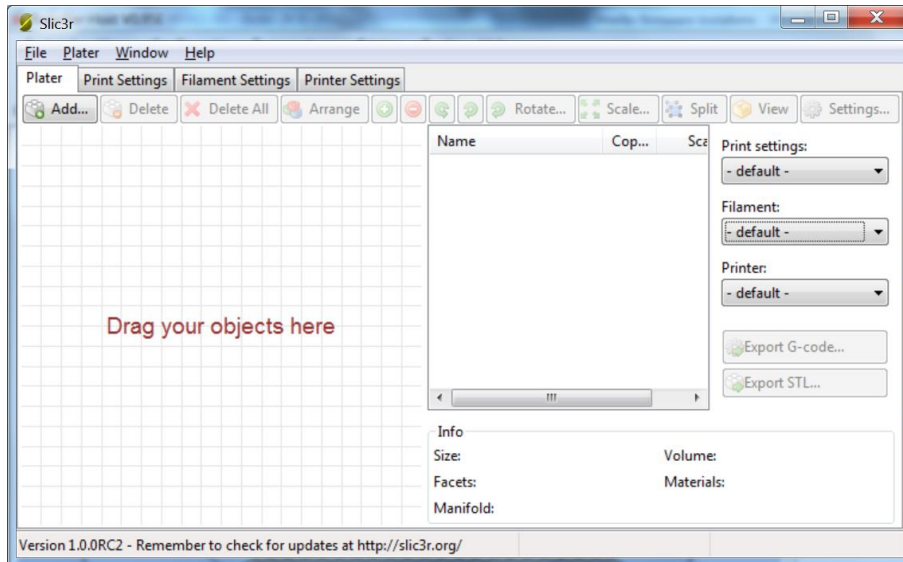
Go to the TAB Slicer and click on the button “Manager”.



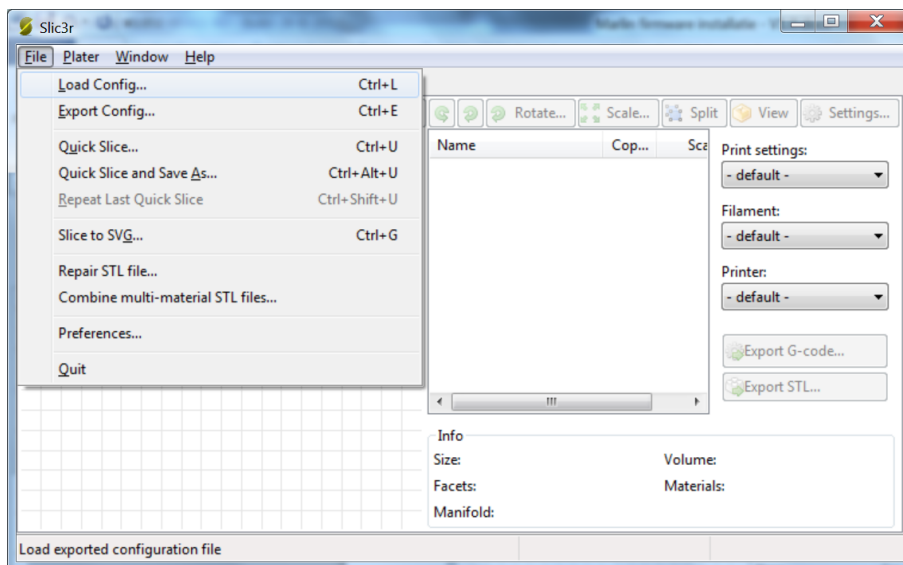
Select “Slic3r” at the left side of the dialog and click “Apply”.



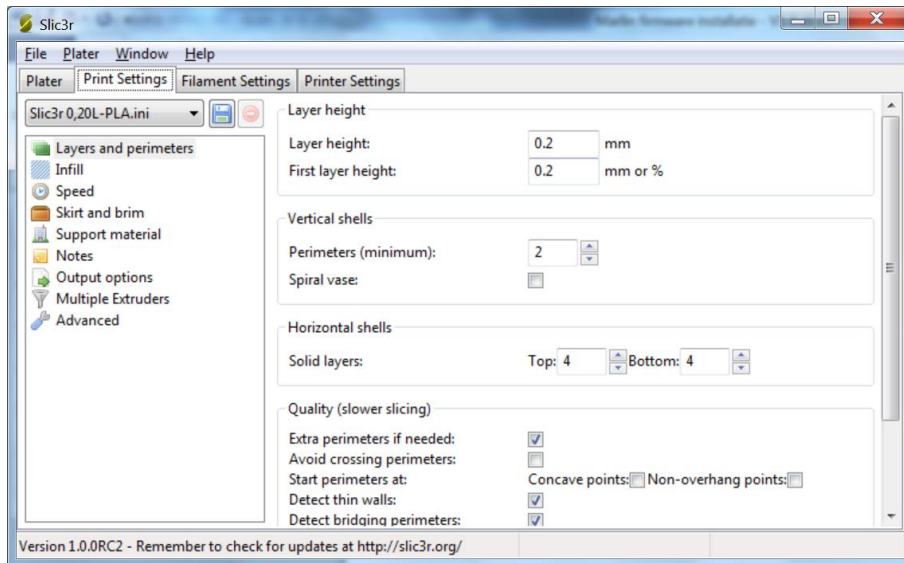
Next click “Configure”.



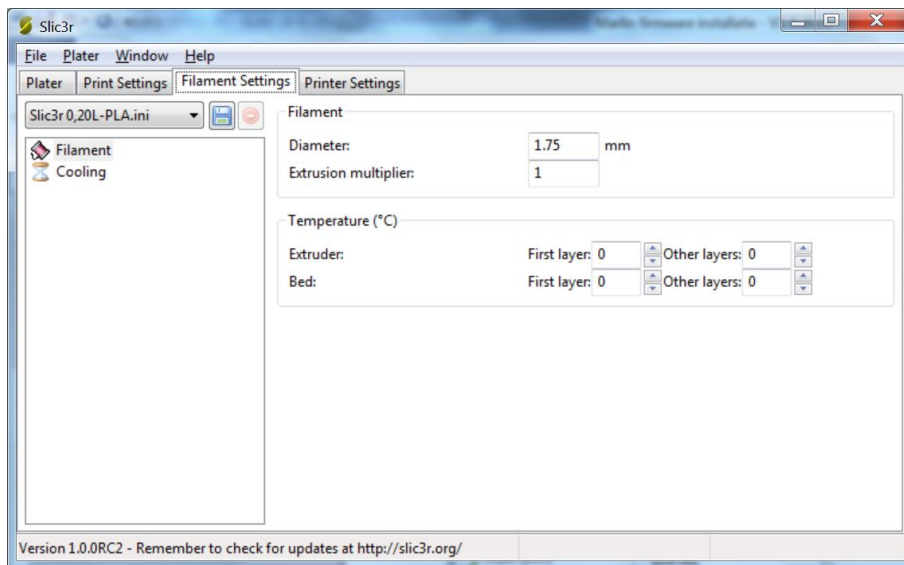
Click “File”; “Load Config” and select the Orca Slic3r Profile.



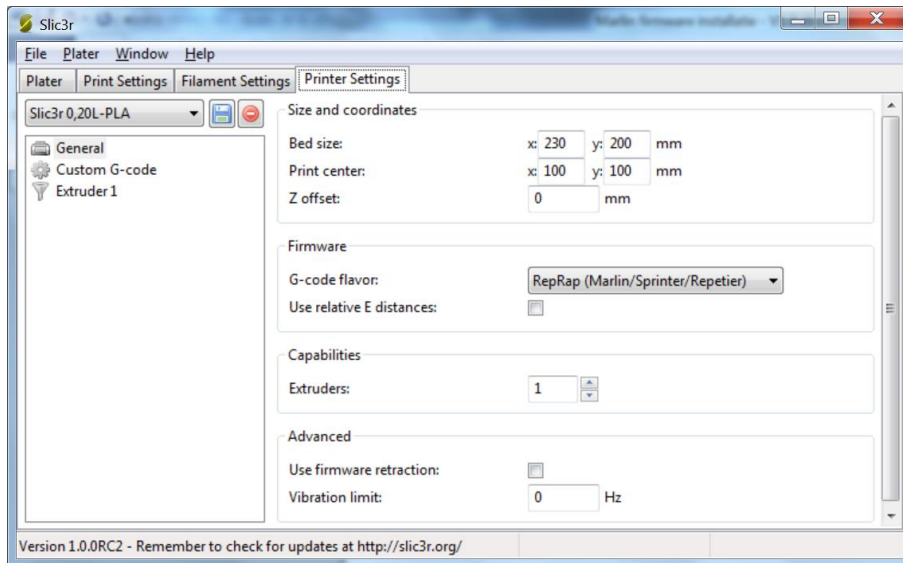
Go to the TAB “Print settings” and save the settings.



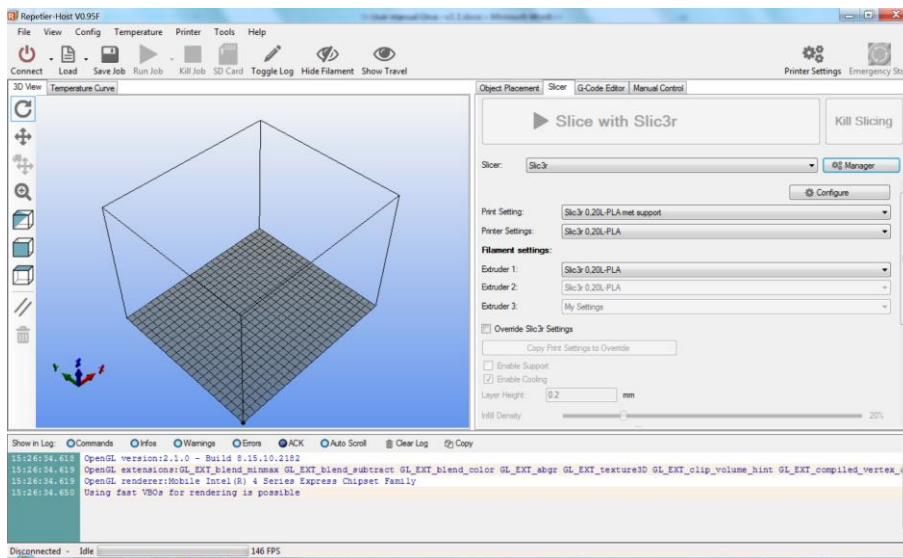
Go to the TAB "Filament settings" and save the settings.



Go to the TAB "Printer settings" and save the settings.

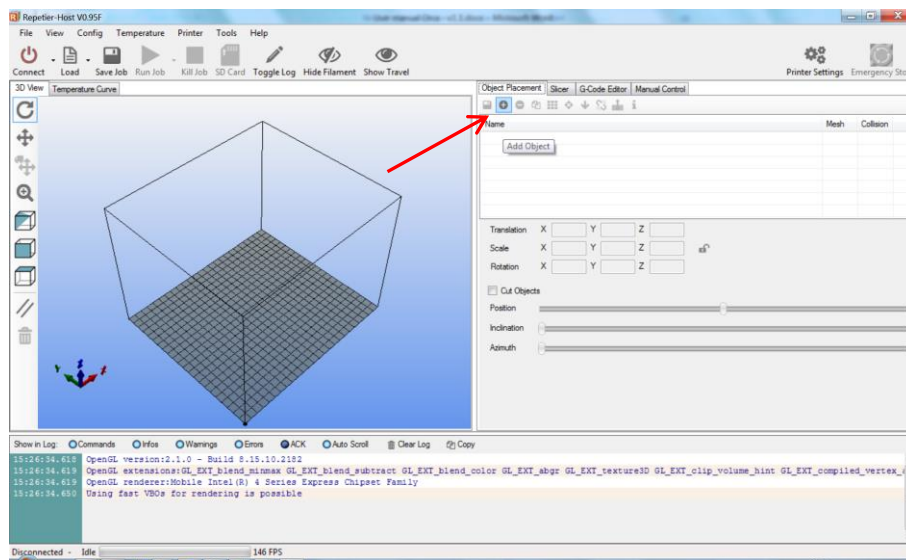


Make sure you also select the new profiles in the Repetier drop down menu's.



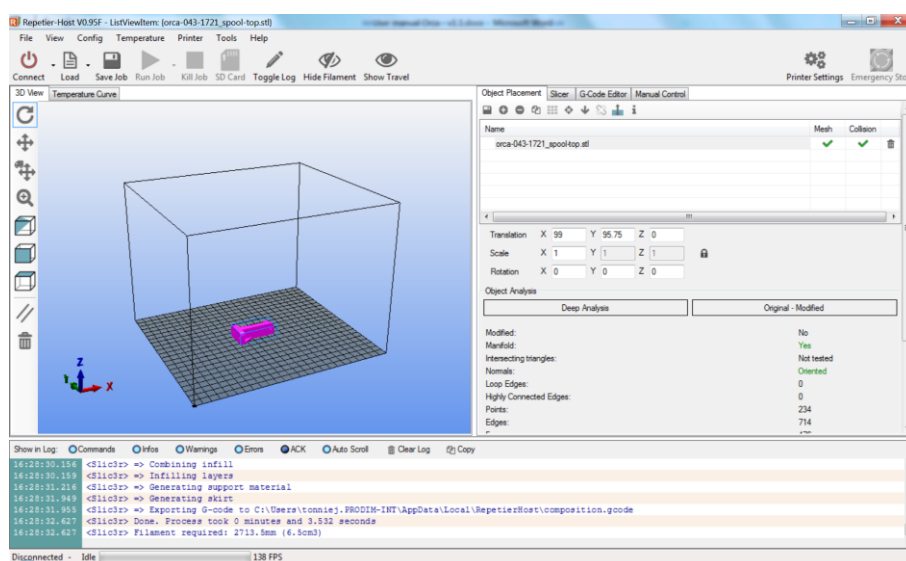
10. Prepare G-code for printing

Before anything can be printed, a 3D file must be exported to or saved as a “.STL” file. These files are suitable for 3D printing. Make sure when saving the file, that the Z-axis is pointing in the right direction (UP).



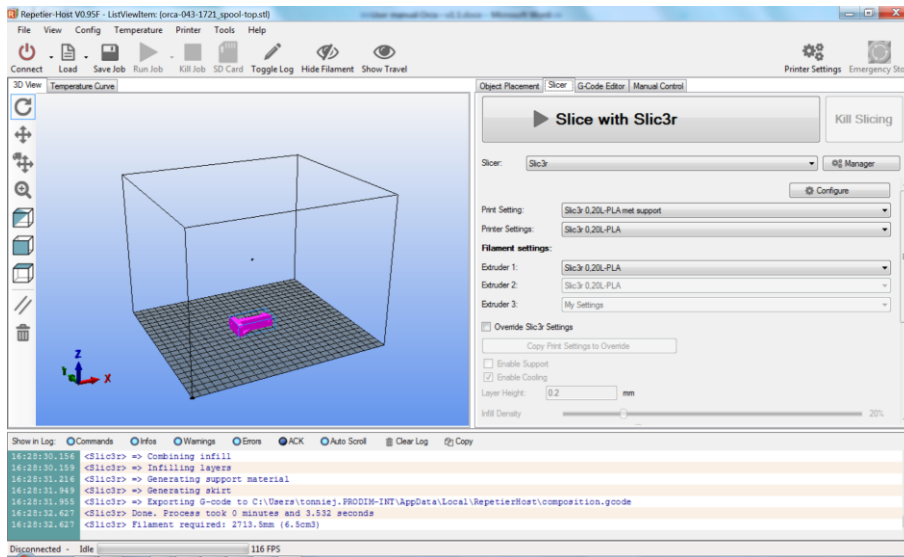
In Repetier STL files can be loaded and will be shown in 3D. Just click on the button “Add object” in the TAB Object Placement and choose the STL file or files. Everytime when a new object is loaded, it will be placed automatically in the screen and will be show in the list of STL objects. It is also possible tot manually move them with the mouse, or use the button: “Center object”.

There is also the possibility to scale the models or place/rotate them with coordinates.

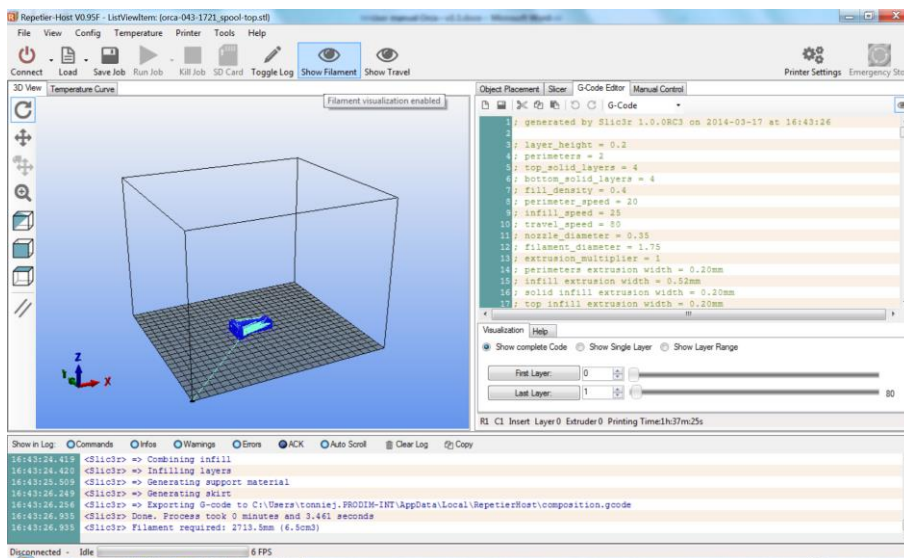




Next step is to Slice the Object. Go to the Slicer TAB and click “Slice with Slic3r”. This can take some time.



After the slicing is done, the program will automatically change to the tab G-code.



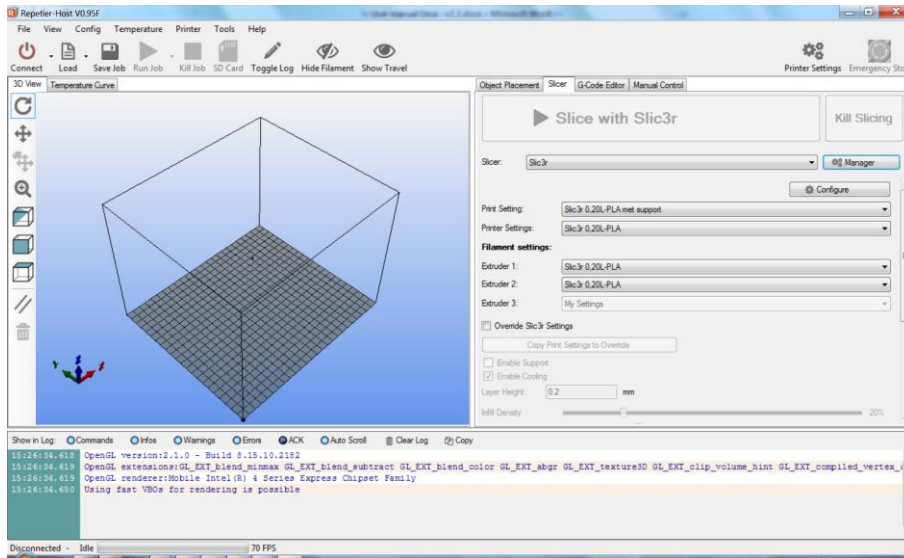
In the object screen a blue “layered” object will show up (if not click on the “Show Filament” button). The dark blue line show the object layer by layer. The light blue lines show the Z-movements (no printing).

You can save the G-code by pressing the Save Job button on top of the screen.

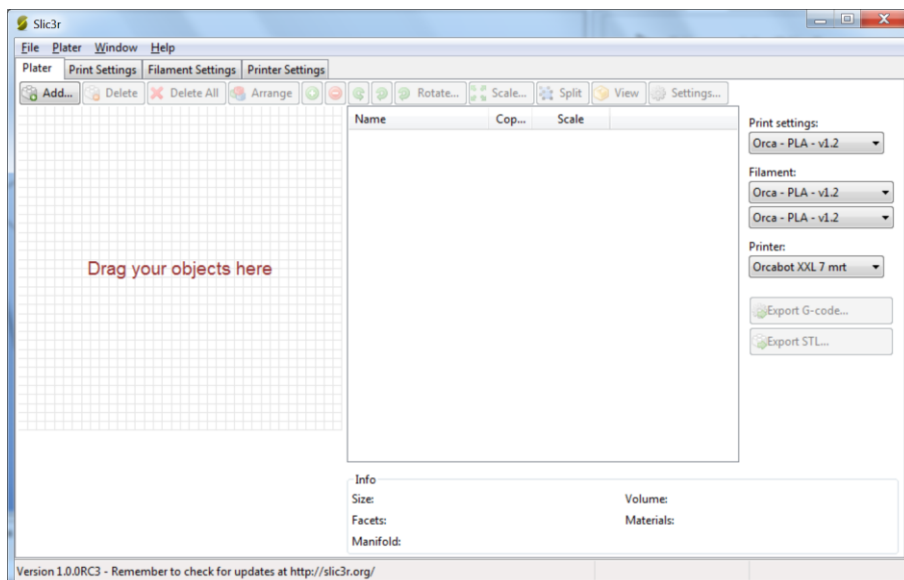
11. Prepare G-code for printing with dual extruder

For printing with dual extruder you need two STL files.

Open Slic3r via the button “Configure” on the TAB Slicer.



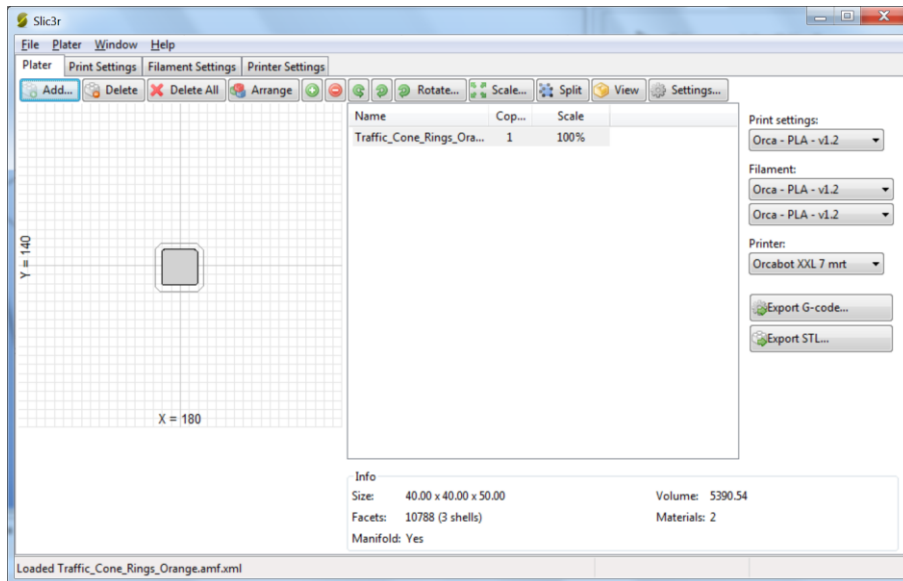
Go to File; Combine multi-material STL files...



After that, select the first file, click Open, select the second file, click open and click cancel.

Next step is to save the new file as an AMF file.

Click on the Add button and select the file. The file will appear on the screen.

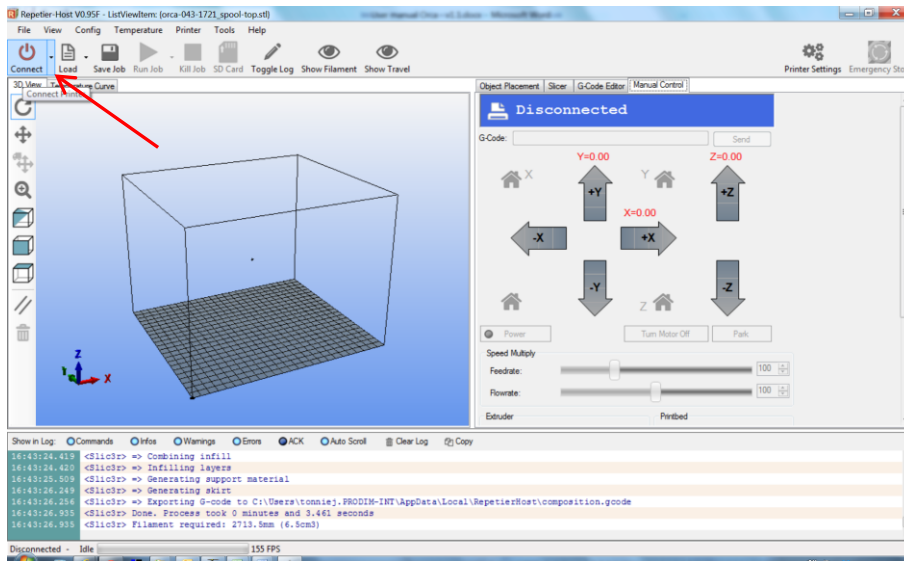


Choose the right Print settings and click Export G-code.

12. Printing

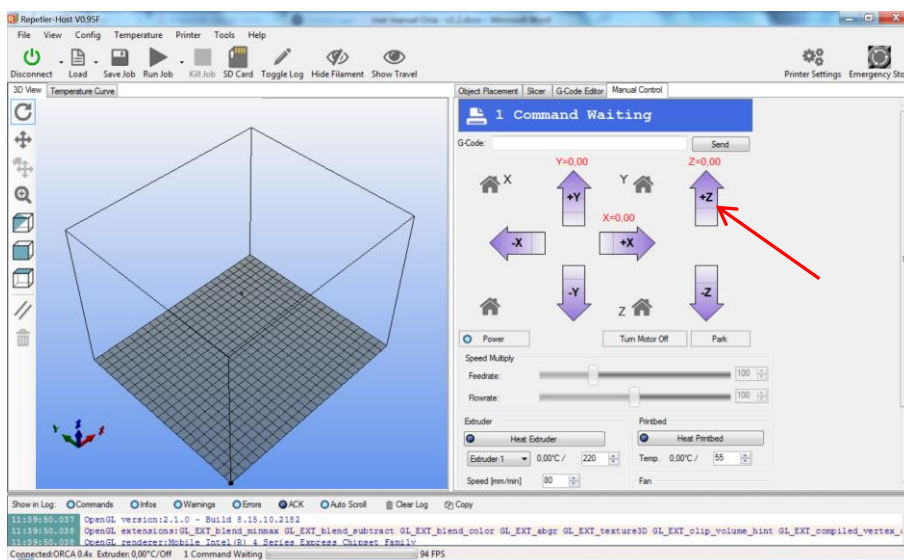
It is always important to start with a clean Printbed. Degrease the Printbed with a towel and e.g. Acetone.

To start printing, the Orca must be switched on and connected to the computer via the USB cable. After that click on the button “Connect” in the top left corner.



The big “Cross” with the arrows are for manual control of the printer. In the corners you can find the home buttons (homeX, homeY, homeZ and homeALL).

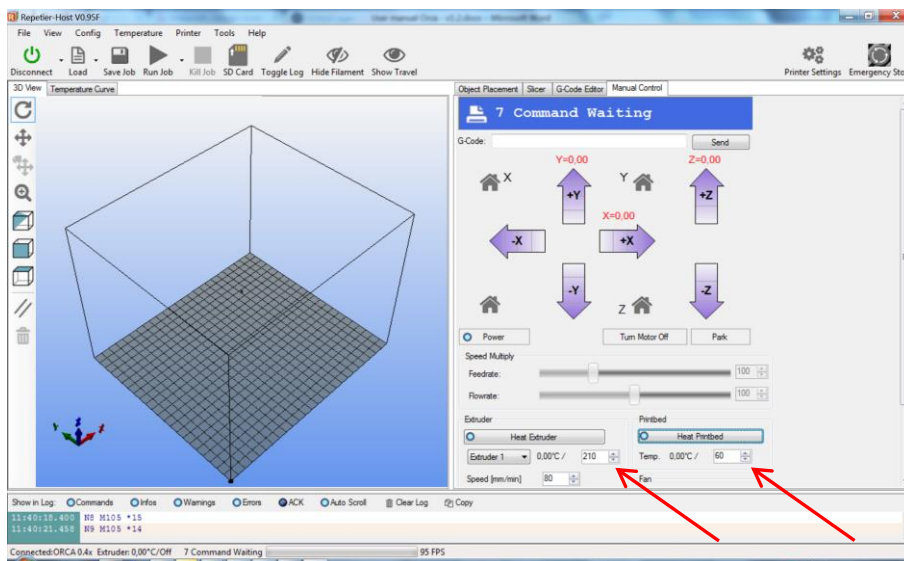
Be careful! Only use homeZ / HomeALL when the Z-is calibrated. Try some arrows and make some moves to make sure that the printer is online and correctly connected.



When the axis are not calibrated yet, please go to the chapter: “Calibration of the axis”. Otherwise you can proceed this chapter.

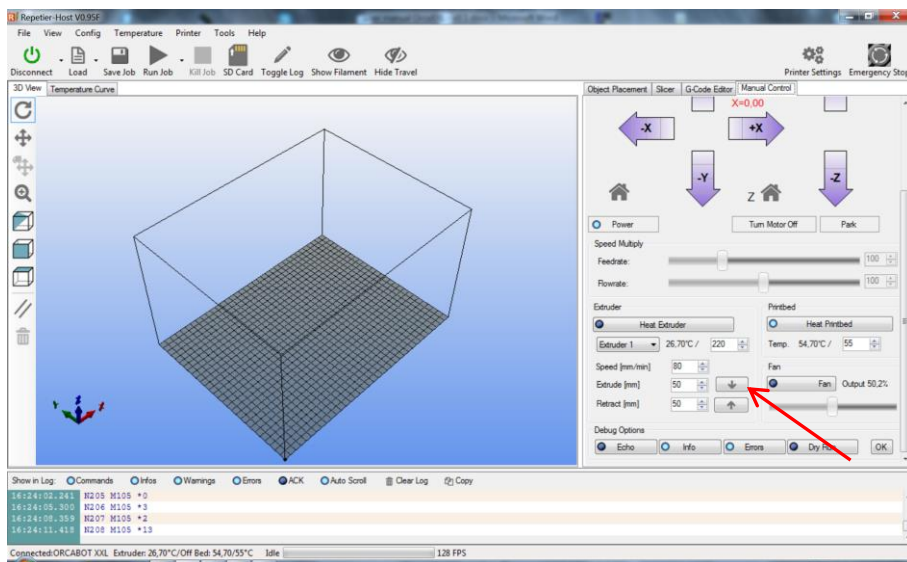
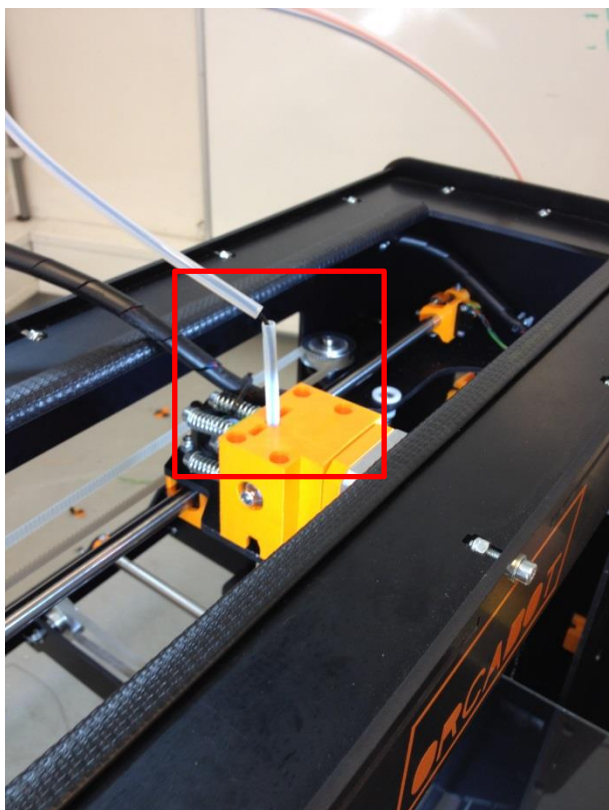
The next step is to set the temperature for the Extruder and the Printbed. First fill in the requested temperature and after that click on the button “Heat Extruder” and “Heat Printbed”. The temperature values in grey are the current temperatures.

Always wait untill the right temperature is reached before continuing!



For temperature settings, please go to www.orcabot.com.

If the temperature is reached, the printer is almost ready for printing, but first manually feed the extruder with some material.



Click in Repetier on the button Extrude for e.g. 50mm. Press the material softly and slowly in the extruder until the extruder takes the material. Feed it until a good amount of material is flowing through the extruder underneath. After this, use a tweezer to remove the material.

Warning: the heater is very hot, so don't touch it with bare hands!

Warning: never touch the printer bed or extruder with bare hands!

Always use tweezers or pliers to remove prints from the bed. If the bed is touched with bare hands, it's possible that the bed lose its cohesion. If so, then degrease the bed with acetone.

If this is done, press "Run Job".



While printing, it's always very important to check the first layer and amount of printed material.

With this wheel on the Z-axis, the height of the bed can be changed (this is only useful for the first layer while printing).

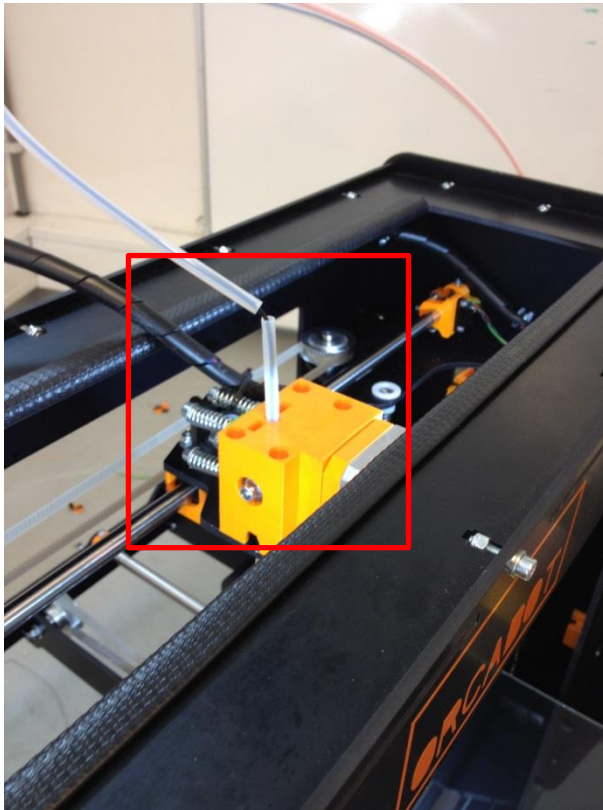
Check the line of material. If it's too thin turn the wheel clockwise to raise the bed a bit. Do this until the lines are completely closed. If the lines are too thick and the material is really

"sweeping up", then turn the wheel counter clockwise to lower the bed a bit.

After printing, the machine will automatically cool of and lower and move the bed away from the extruder head. Wait for the machine to cool off and then remove the printed object. Reminder, never touch the bed of extruder with bare hands. Use pliers or tweezers for removing parts (not manually, otherwise you need to degrease again).

13. Replacing material

If the spool with material is (almost) empty, or when another material or color is wanted, it is time to replace the spool.



Switch on the printer and run Printer interface. Switch on the heater and wait till it has reached its temperature.

Try to leave min. 1 cm filament sticking out. After that, Retract the extruder via the Retract button in Repetier and remove the old filament which was in the extruder. After that insert the new filament.

