

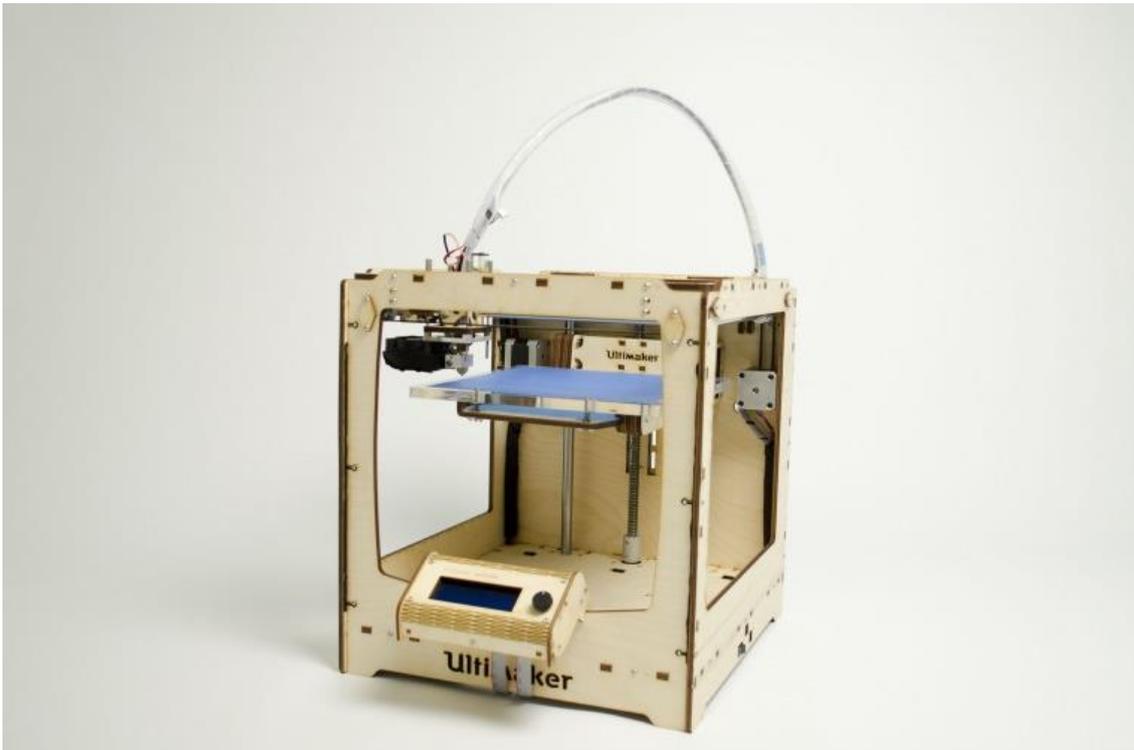
HOW TO USE AN ULTIMAKER

Written by Marti Bertran

In this section:

- How to prepare an Ultimaker from printing
- How to run a print job
- What is in the menu with the blue display
- Summary

This is a quick user manual for using the Ultimaker 3D printers available in the DTU Additive Manufacturing Fablab. However if you are new to the Fablab and to the machines this is not enough to operate them. Therefore write an email to 3dprint@mek.dtu.dk and arrange an appointment for a crash course.



WARNINGS

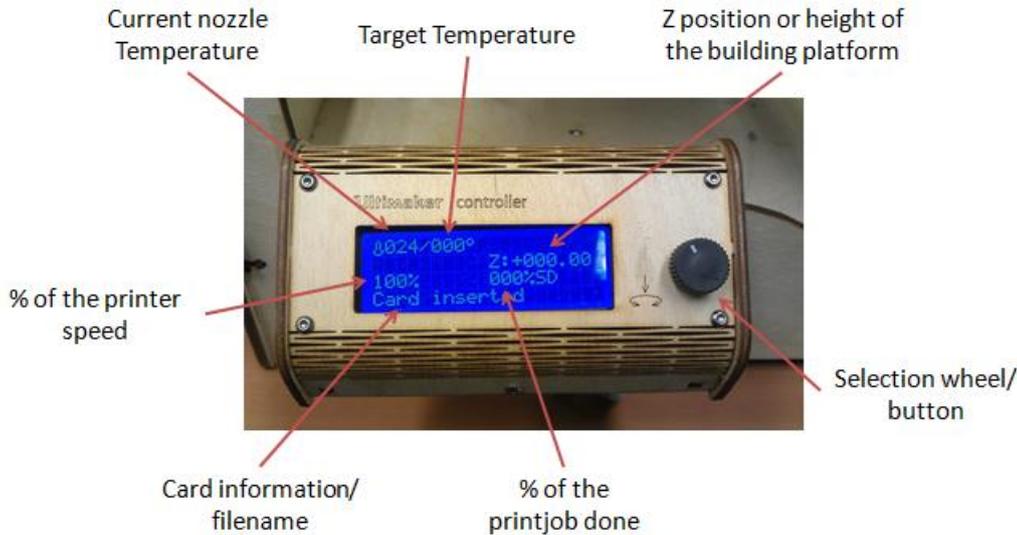
- Do not proceed to use the printer If you have any doubts after reading this manual, ask the technicians before doing anything.
- If you notice that one printer is broken or is not working properly, do not try to fix it, report what is happening to the technicians, they will help you to solve the problem.

BEFORE STARTING

- Make sure you have generated a GCODE of your stl files, and saved it in the printer's SD Card.
- Change the tape on the building platform if damaged
- Sand the surface to get rid of the imperfections
- Clean with Isopropanol solution to remove the dust from the top of the plate

HOW TO USE THE MACHINE

HOME MENU INFORMATION

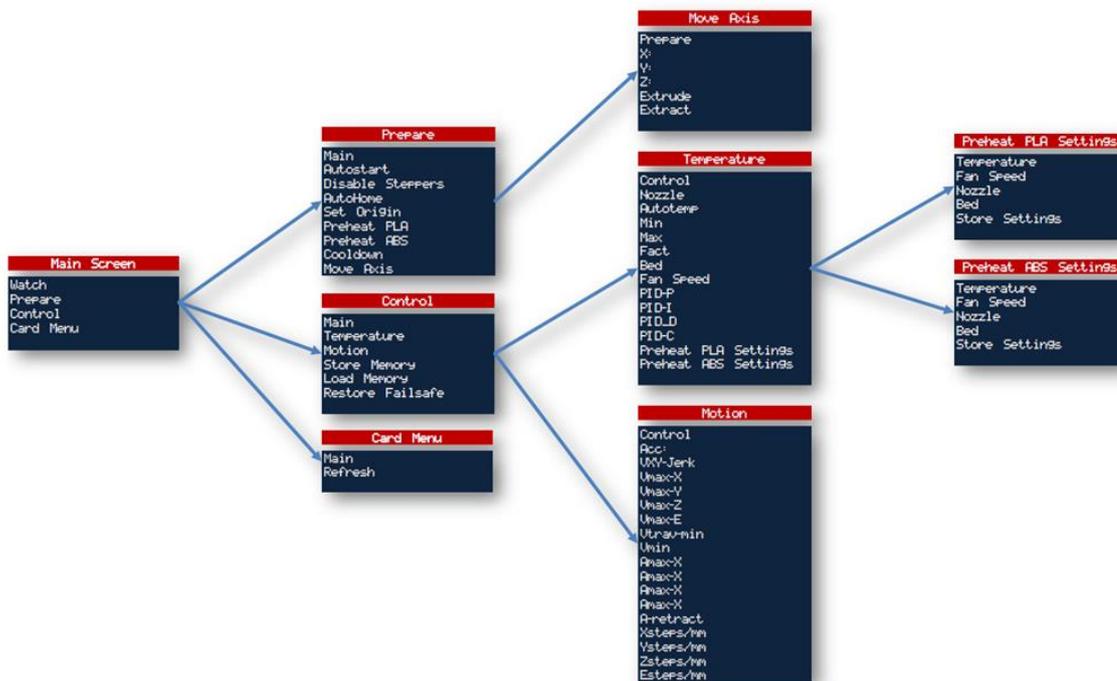


It is possible to change some of the operative settings of the machine directly from the display placed in front of it. On the LCD is possible to find information relatives to:

- **Current nozzle temperature:** Actual temperature of the printer nozzle.
- **Target temperature:** is the temperature that the Ultimaker is aiming for. Without a print command this is 0. When for instance, preheating PLA, this value will become 180°C, and with a print command for example 220°C. As a result you will see the first temperature rise, and fluctuate around the targeted temperature.
- **% of printer speed:** Underneath the temperature the speed shown as a percentage is displayed. In this case 100%. When a print has been initiated, the 100% will get more meaning based on the settings of the Gcode. For example if you set your print speed to 40mm/s, 100% is 40mm/s. If the speed is increased to 120% , the speed changes to 48mm/s. This involves all speeds, perimeter, infill and minimal layer speed.
- **Card information/ filename:** If it is 'Card inserted' it means the SD-card is inserted, and it is accessible. When card is removed the text 'Card removed' will be displayed.
- **% of the print job done:** 000%SD keeps track of how far you are into you print. This means you haven't started, 100% means you print is finished. This indication is based on layers.
- **Z position or height of the building platform:** is the current position of the Z-stage. When you are printing your Z stage will slowly move down with tiny steps. This number indicates how far it has been lowered in mm.

LCD MENU TREE

By pressing the wheel you will enter the main menu. A graphical representation of the information are displayed in the figure below.



Prepare

Via Prepare you will proceed to the next sub-menu where you can find a variety of settings that can be used to prepare a print.

- **Main:** go back to the main menu
- **Autostart:** If there is a file named auto0.g on the SD-card, it will automatically begin printing this file when you turn on the Ultimaker.
- **Disable steppers:** With this function the steppers motors can be disabled. These allow you to move the print head and the building platform without the resistance from the motors. By sending a command to use the motors, for example, 'Move Axis', they will be activated again.
- **Auto home:** nozzle is sent to the home position that is the front left corner.
- **Set origin:** Set origin will make the current position to the 0,0,0 so: home.
- **Preheat PLA:** With Preheat PLA the nozzle will heat up to 180°C
- **Preheat ABS:** With Preheat ABS nozzle will heat up to 240°C (however no ABS is used in the Fablab).
- **Cooldown:** Set the target temperature to 0
- **Move Axis:** With Move axis another sub menu is open which allows moving the axis around. Move the X, Y and Z axis by clicking on the X, Y or Z in the menu and rotate the button. By clicking extrude a command to print 1 mm of filament is given. This will only work when the nozzle is at least 180°C hot.

Control

- **Main:** go back to the main menu
- **Temperature**
 - **Nozzle:** set nozzle to a certain temperature.
 - **Fan speed:** change the speed of the fan from 0-255

- **Motion**
 - **Acc:** The standard is 3000, it allows to adjust the acceleration speed.
 - **Vxy Jerk:** Vxy Jerk is the acceleration of the acceleration
 - **Vmax x:** The maximum speed in mm/s
 - **Vmax y:** The maximum speed in mm/s
 - **Vmax Z:** The maximum speed in mm/s
 - **Vmax E:** The maximum speed in mm/s
 - **Vtrav min:** This is the minimum distance to travel
 - **Vmin:** The minimum speed in mm/s
 - **Amax x:** The speed in which the motors can accelerate.
 - **Amax y:** The speed in which the motors can accelerate.
 - **Amax z:** The speed in which the motors can accelerate.
 - **Amax e:** The speed in which the motors can accelerate.
 - **A retract:** The acceleration speed in which you allow retraction to operate
 - **X steps-mm:** The accuracy the steppers can make the motors move in.
 - **Ysteps mm:** The accuracy the steppers can make the motors move in.
 - **Zsteps mm:** The accuracy the steppers can make the motors move in.
 - **Esteps mm:** The accuracy the steppers can make the motors move in.
- **Store memory:** save settings before continuing changing parameters
- **Load memory:** Load the previously saved parameters
- **Restore failsafe:** Restore failsafe will reset the UltiController to its original settings. All the settings previously made will be deleted.

Card Menu

When accessing this menu all the Gcodes on the SD-card will be visible. By clicking one of them the machine will start to print it. If the nozzle is not heated yet, it will first start heating and then starting the print.

Tune (Only available while printing)

When a print has been initiated 1 the *Prepare* is replaced by *Tune*. In the tune-menu it is possible to live tune the printer. Meaning parameters of the print can be changed, while printing.

- **Speed:** The printing speed will increase/decrease. This can also be done faster by just rotating the wheel in your home menu. If you change the speed, it will have effect on the perimeter, infill and travel speed.
- **Flow:** Flow allows increasing or decreasing the amount of filament that is being extruded.
- **Nozzle:** change the temperature of the hot end.
- **Bed:** This has no meaning in the current set-up. When a heated bed is officially released by Ultimaking Ltd this setting will have a meaning.
- **Fan Speed:** You can change the speed of the fan from 0-255. In the slicer you can set if you want the fan on, off or with an interval.

PROCEDURE

1. Turn on the machine using the On/Off switch in the right side of the printer
2. Insert the SD card containing the Gcode of your part in the controller slot
3. Using the button of the controller to navigate through the menu, select "preheat PLA", the route is: Main menu → Push the button → select "Prepare" → Push → select "preheat PLA" → Push
4. Now the Nozzle is heating up. Wait until the temperature has reached 180°

5. Extrude a bit of plastic in order to renew the material in the nozzle before starting the printing the route is: Main menu → Push → Prepare → Push → Move axes → Push → Extrude
6. Remove the strings of filament from underneath your hot end so it won't interfere with your first layer
7. Select the Gcode file to be printed, the route is: Main menu → Push → Card Menu → Push → Yourfilename.gcode → Push
8. The printer will wait until the nozzle reaches the target temperature set on the Cura
9. Wait until the first layer has been printed successfully, if you notice that something is going wrong, Select "Stop printing" and ask the technicians for help.