Using the Ultimaker 3D-Printer

The Ultimaker is an FDM type 3D-printer. FDM stands for Fused Deposition Modeling which means that the printer prints by extruding layers of plastic on top of each other. By the heat of the printing nozzle and just extruded plastic the layer underneath melts a little bit and the two. When the two layers cool down they are ‘fused’ together. You could mimic the same principle using e.g. a glue gun.

The model

To start printing you’ll first need a 3D file. This file can be created using your favourite 3D modeling program like Sketchup, SolidWorks, Rhino etc. The file needs to be loaded into Cura, the program that slices your model into layers. Cura accepts the following formats:

- binary STL
- DAE
- OBJ
- AMF

The units cura expects your model to be in is **millimeters**.

Make sure your model is watertight. If not the result might not be what you expected. To check and fix your model you can use a program like Meshlab.

Slicing using Cura

Now your model should be made suitable for printing and transformed into GCode, the language the 3D-printer talks. You can do this with several programs, we will use Cura. If you don’t have cura you can download the latest version from http://software.ultimaker.com/. During the setup make sure you choose *Ultimaker Original*.

Newbies / never printed before

- load your model with File -> Load model file
- If not already switch to quickprint with Expert -> Switch to quickprint
- Now you can select high, medium or low quality. Medium is good for most prints. Choosing high results in finer details but will increase the duration of your print considerably.
- Under the Material settings make sure material is set to PLA and Diameter is set at 2.85.

- Now you can save your model as GCode to an SD card with File -> Save GCode. Make sure you add .gcode at the end of the filename.

Intermediate/Experts/want the cool stuff

- After loading your model with File -> Load model switch to advanced mode with Expert -> Switch to full settings. You now have considerable more options to tune.
- If you are unsure about a specific setting you can hold your mouse over it. A help text should appear and as well indicate some sane default values.
- Some reasonable default values:

Settings	**Description**	**Reasonable values**
Layer Height	Height of each printed layer	Ok 0.2 Very fine 0.1
Wall thickness	Thickness of the outer shell	0.5 - 1.2
Bottom/Top Thickness	Thickness of the bottom and top layer. This should be a multiple of the layer thickness and can not be less than the layer thickness.	0.4 - 1.0
Fill Density	The percentage the object will be filled. 0% means no fill at all (hollow). 100% is completely filled. Suggested range for a filled object is between 20 and 40%. 100% infill for a solid object is not recommended as the end result will not be very pretty (think of colouring a drawing with a big marker, there will be *overshoot* at the edges).	20% - 40%
Enable retraction	To prevent wires between travel points the printer needs to retract the filament a little bit. You can turn this on with this option.	on
Speed	Printing speed in milimeters per second. The value needs experimentation and depends on filament, temperature and model. It is possible to change this on the fly by turning the knob on the Ulticontroller.	80

Temp	Printing temperature. Depends on your model, printing speed and filament. A good start value is 210 or 220 degrees Celsius. If your model has overhang printing colder and slower can improve the result.	PLA: 180 - 220 ABS: 230 - 240
Support Type	If the model has overhang with angles larger than 45 degrees you should print support to be able to print this features.	(geen)
Platform adhesion type	Raft: Thick layer of filament. Used to level irregularities in the platform. Usually not necessary. Brim: prints several contour lines attached to the model. This can help with adhesion to the platform and *bending* of larger surfaces. The amount of brim lines can be set in the expert menu.	uit

Removing the filament

- Heat up the printer (on the Ulticontroler select: Prepare -> preheat PLA). Wait for the nozzle to reach a temperature of >=180 degrees. **Never try to remove the filament with a cold nozzle. PLA works as superglue when cooling down. Trying to remove it cold will break the extruder.**
- Turn the feeder wheel a little bit counter clockwise. This will compress the filament a bit in the nozzle.
- Directly turn the feeder wheel clockwise for several rotations. This should go easy.
- Open the filament clamp on the feeder. If you don't know how to do this ask one the Station instructors.
- Pull the filament back.

Inserting the filament

- Heat up the printer (on the Ulticontroler select: Prepare -> preheat PLA). Wait for the nozzle to reach a temperature of >=180 degrees.
- Open the feeder and push the filament into the bowden tube.
- Push the filament all the way thru until you feel some resistance.
- Close the filament clamp
- Turn the feeder wheel counter clockwise until you see the right colour filament being extruded out of the nozzle.

3D printing is not as simple as it seems. To make a good quality print you need to try and experiment a lot with the different settings and gain some intuition of what will work and what not. EXPERIMENT!!