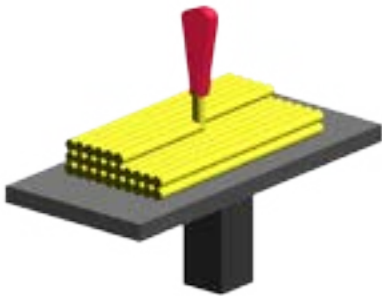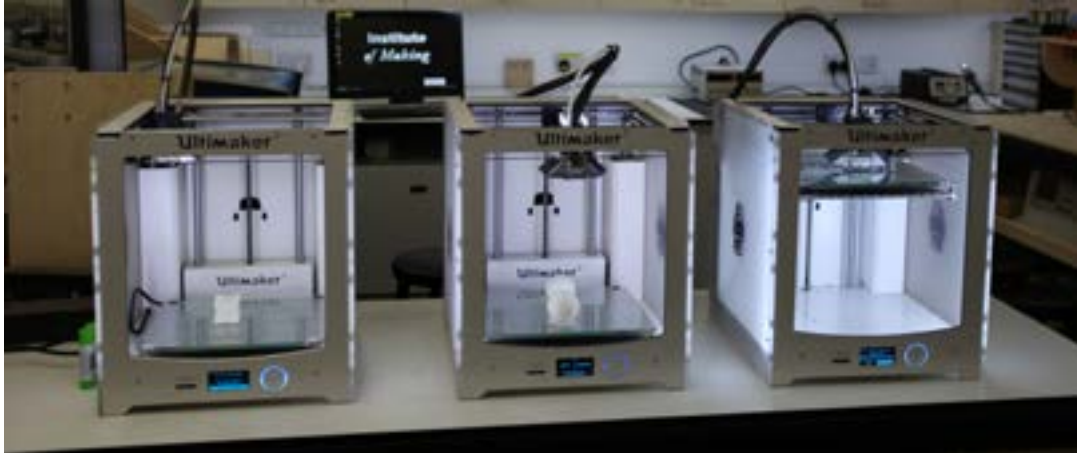## Intro





We have 3 brilliant Ultimaker 2 3D printers you can use to create 3D models. The printers are currently loaded with white filament called Polymax PLA which is free for you to use. The Ultimakers use a technology called Fused Filament Fabrication (FFF) or Fused Deposition Modelling (FDM). This technique works by moving a printhead and depositing melted filament on a build plate to slowly build up the object.

## Basic Workflow

**Create or download 3D model -> Slice into layers -> Print file**

Create (please see the document on CAD) or download a 3D model in format .stl or .obj. There are vast amounts of 3D models online, www.thingiverse.com is a good place to start but you will find many more by searching the web (other good sites are www.123dapp.com/gallery and www.youmagine.com). The Ultimakers can print any models up to size 223mm x 223mm x 205mm.

Slice your model using slicing software, we use Cura which can be freely downloaded for both Mac and PC (ultimaker.com/en/products/cura-software). The software slices your solid 3D model horizontally into thin layers to enable it to be printed.

## Importing

To import a 3D model into Cura either go to File -> Load model file or simply drag your 3d model file and drop it inside the box on the screen.
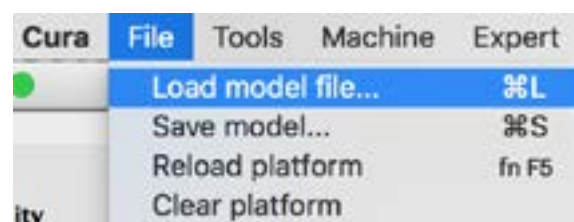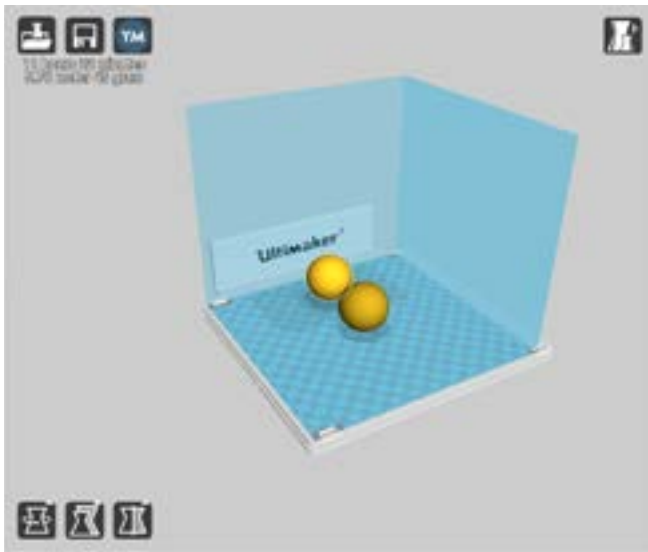
Check your model has turned yellow, this means it fits inside the bounding box and is small enough to be printed. Your screen should look something like this.

In the bottom left hand corner of the program there are some tools for manipulating the size and position of your object inside Cura. You can also click and drag to move objects around on the print bed, they will turn grey if they you place them outside the printable area.

It is a good idea to think about the orientation of your print to give it the best chance of printing without any problems. If a print is too large to fit in the print area you can scale it down with the scale tool.

Please note, it is best practise to make any changes to your model, including size and orientation changes, inside CAD software as Cura will not allow you to save any changes to your model.

Once you have imported your model there are a number of options to set for your print. With all of options you should be trying to optimise quality vs print time.

## Settings - Quality
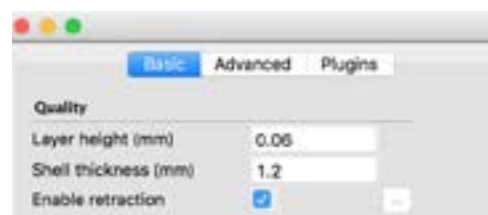
**Layer height (mm)** - Range: 0.06mm - 0.25mm

The height of the layers the model is sliced into, 0.06mm will result in finer, higher quality prints and 0.25mm prints will be much faster but will be less detailed.

**Shell thickness (mm)** - Range 0.4mm -

This sets the thickness of the outer shell, or skin, of your print. This value needs to be a multiple of the printer's nozzle size (0.4mm). A shell thickness of 0.4mm will print a shell one layer thick, 0.8mm two layers thick and so on.

**Enable retraction**

This tells the printer to retract the filament from the nozzle while moving between different areas of the print. This helps to reduce stringing (strings of filament trailing between parts of your print).

## Settings - Fill

| Fill | |
|---|---|
| Bottom/Top thickness (mm) | 0.6 |
| Fill Density (%) | 20 |
| **Speed and Temperature** | |
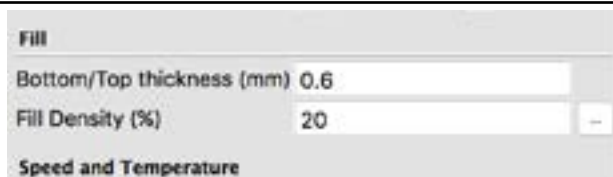
Most 3D models we download or draw in CAD are solid shapes. When it comes to printing a model, filling the whole of the inside of an object can increase the print time dramatically. For this reason Cura makes it easy to make your object hollow, there are two variables you can set to change how hollow your print is.

### Bottom/Top thickness

This value affects how many solid, complete layers the Ultimaker will print above and below the hollow internal part of the object. It is a good idea to use a value which is a multiple of your layer height.
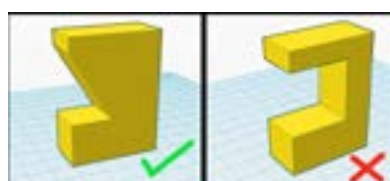
### Fill Density (%) - Range 0% - 100%
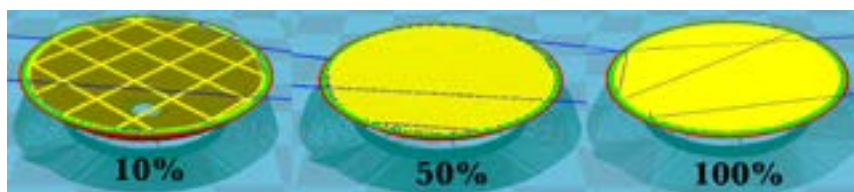


The fill density changes how solid your object is.

A fill density of 0% will simply print the outer shell of the object, 100% will result in a completely solid print. Anything in between will create an internal lattice inside your print, the higher the value, the smaller the gaps between the lines in the grid.

### Print speed (mm/s) - Range up to 150 mm/s

High print speeds will decrease print time but increase the chance of the print going wrong. A value between 80-100 mm/s usually works well.

## Settings - Support



| Support | | |
|---|---|---|
| Support type | Touching buildplate | ⌄ |
| Platform adhesion type | Brim | ⌄ |

Select the support structure you would like for your print. A model will need supports if it contains overhanging sections as these will droop or fail to print without support. You can chose from "Touching build plate" which will put supports in wherever there is nothing between your overhang and the build plate and "Everywhere" which will also put supports in where your model overhangs itself.

As a general rule if your 3D model has sections which overhang more than 45 degrees it will require support structure.

## Settings - Platform Adhesion

| Support | | |
| --- | --- | --- |
| Support type | Touching buildplate | |
| Platform adhesion type | Brim | |

Select a Platform adhesion type, this can be "none", "brim" or "raft".
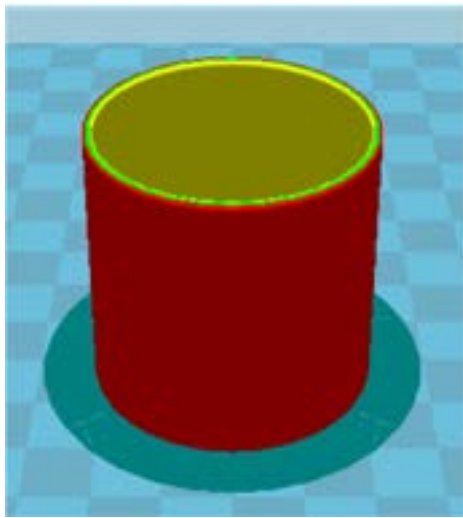


A brim, like a hat brim, will print a base layer slightly larger than the footprint of the object you are printing.

In this image the blue/green section is the brim.

A brim helps a print stick to the build plate, it also makes it easier to peel off once the print has completed.

A raft will print a more substantial base for your object, this can help with the structural integrity but can make it tricky to remove your print from the build plate. If the object you are printing has a small surface area touching the build plate you may want to consider using a raft to provide a strong base to build on.
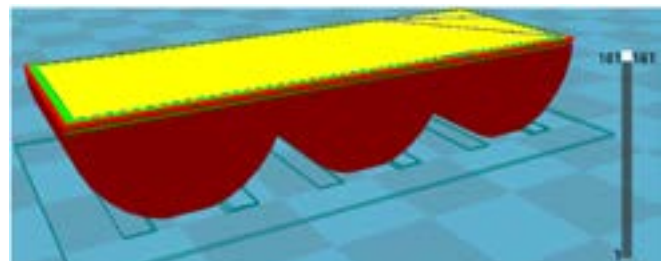


If your print has a large surface area touching the build plate it may not be necessary to include a brim or a raft.

## Exporting

To export your .gcode file to an SD card. Plug the card in the side of the computer.



Once the card is inserted the Save toolpath button in Cura will change from saving locally to saving to the card.
It is a good idea to use a name you will be able to find easily.

Pressing the button will generate your .gcode file. .gcode files tell the printers what to print and how to print it using the settings you have provided.

## Print

Put the card in an Ultimaker, navigate to print and finally find your file and select it. Press in on the scroll wheel to select menu options.





## Rules

Don't use more than one printer at once.

Don't put your hand inside the printer when it is printing or before it has finished cooling down.

Try and be considerate of other members, if you have a print that is going to take 2 days please try and arrange to set it off on a friday afternoon so it can print over the weekend.



Please note: The printers are quite temperamental, our printers are in use fairly constantly throughout the day and as a result they often break down, nozzles clog up and other problems arise. If this happens during your print or you notice one not functioning properly please let one of the technicians know and we will get it back up and running as soon as possible. Happy Printing!