

## Printers & Technology



- We have currently 4 Ultimaker 3D printers that you can use to print your 3D models. One Ultimaker 2, two Ultimaker 2+ and an Ultimaker 3 Extended. The Ultimaker 3 Extended has a dual nozzle which can potentially allow you to print supports with PVA (and dissolved in water). Please see a technician if you want to use the second nozzle as printing with PVA can be very challenging.

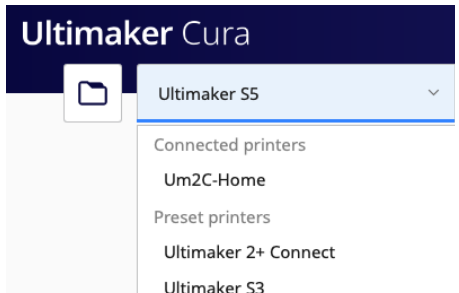
- The Ultimakers use a technology called Fused Filament Fabrication (FFF) or Fused Deposition Modelling (FDM). This technique works by moving a printhead and depositing melted filament on a build plate to slowly build up the object.
- These printers are currently loaded with white recycled PLA filament called Filamentive which is free for you to use up to 100g per print. We don't use any other type of material / colors on these machines in order to simplify their maintenance.
- We also have an experimental 3D printer from RS Components that can be used with different materials such as NinjaFlex or different colours of PLA. This printer works with another software, so please see a technician if you want to use it.

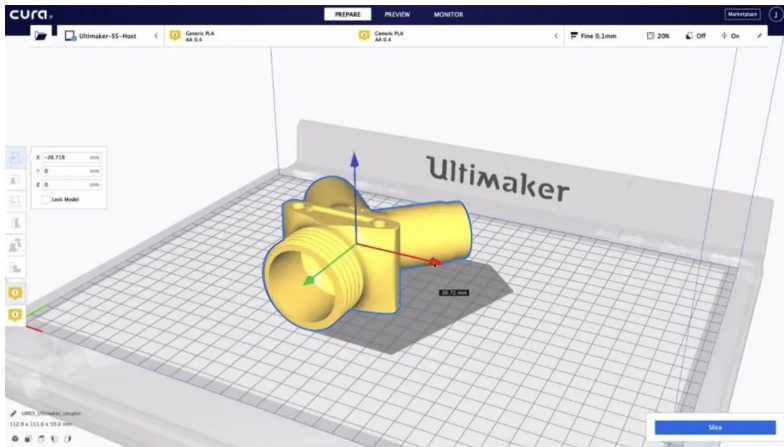
## Basic Workflow

- First create or download a 3D model, then slice your model into layers, then print the file.
- Create or download a 3D model in format .stl, .obj or .3mf. There are vast amounts of 3D models that you can download online. 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](http://www.123dapp.com/gallery) and [www.youmagine.com](http://www.youmagine.com).  
The Ultimakers can print any models up to size 223mm x 223mm x 205mm. (or 215mm x 215mm x 300mm for the 3 extended)
- 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](http://ultimaker.com/en/products/cura-software)). The software slices your solid 3D model horizontally into thin layers to enable it to be printed.

## Open 3D files

- The easiest way to open 3D models is by the 'open file' button. The button is in the top left corner and is the first button in the stage menu. Select the model you want to prepare for printing and open it. The model will now be loaded and shown on the 3D viewer.





- 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.

- On the left 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 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 the options you should be trying to optimise quality vs print time.

## Print Settings

- The rightmost button in the stage menu contains the print settings panel, which includes all the settings that define the printing strategy. By default, it opens in the recommended mode. This mode is ideal if you want to do a quick print with optimized printing profiles. There are a few parameters you can set:

- 1. Print settings panel.** This panel shows the current printing strategy at a glance, click it to view the entire panel.
- 2. Print profiles.** The available profiles for the current configuration are visible. Some Ultimaker 3D printers offer intent-based profiles, fine-tuned to meet specific needs.
- 3. Infill.** The infill slider can be used to easily set the overall model strength.
- 4. Support.** Enable or disable automatically generated support structures, with either of the available extruders, to get reliable and successful prints when necessary.
- 5. Adhesion.** Enable or disable additional adhesive printed parts, the type of adhesion is automatically set by the print profile.
- 6. Custom mode.** This will open the custom mode to manually finetune printing profiles.



**Custom Mode****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 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).

**Fill**

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.

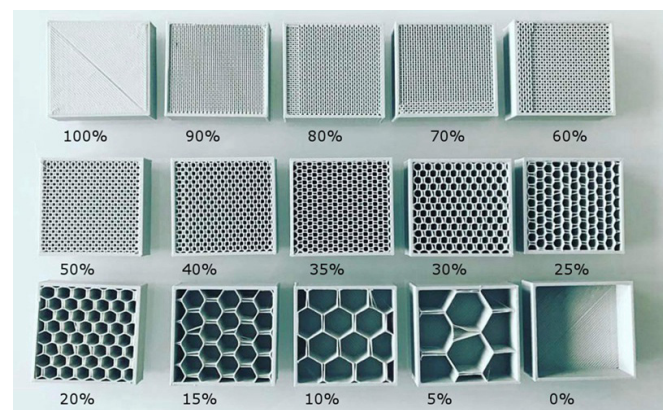
**Bottom/Top thickness**

This value affects how many solid, complete layers the Ultimaker will print above and below the hollow internal part of the object. Use a value which is a multiple of your layer height.

**Fill Density & Pattern**

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. Infill comes in different shapes, sizes and patterns, each with its own pros and cons between print time, structure strength, and material usage.

Note: infill less than 10% can lead to problems.

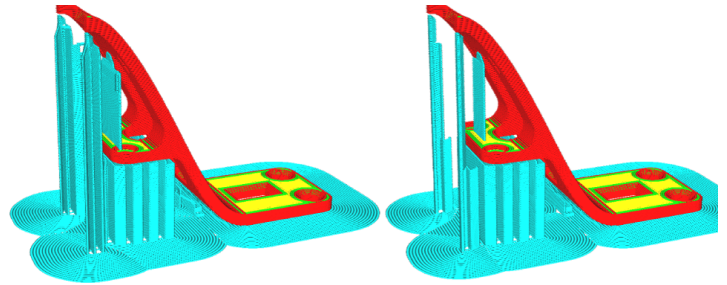
**Print speed (mm/s)** - Range up to 150 mm/s

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.

**Support**

Select the support structure you would like for your print. A model will need supports if it contains

overhanging sections as these will drop or fail to print without support. You can choose 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. Cura will display overhangs in red when a model is loaded.

### Adhesion

Select a platform adhesion type, this can be “none”, “brim”, “raft” or “skirt”.

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

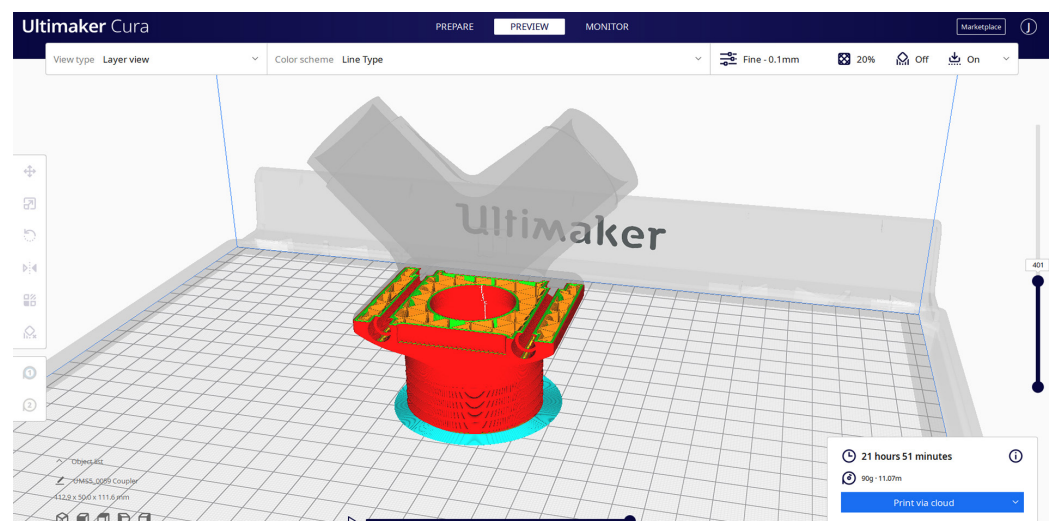
A raft will print a more substantial base for your object, this can help with the structural integrity but can make it tricky to remove your print from the build plate. If the object you are printing has a small surface area touching the build plate you may want to consider using a raft to provide a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

### Slice and Preview

Now that the printer type, configuration and print settings are in place, it's time to slice the model with the button in the bottom right corner. When the process completes it will immediately show a 'preview button', click it to go to the preview stage.

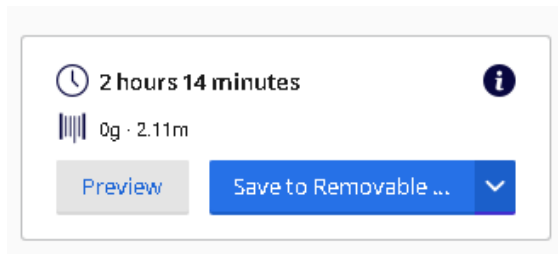
This view is ideal for seeing what the inside of your print will look like and how it will be printed. Use the layer slider on the right and simulation view to check important parts of your 3D slice.





**Load**

Once you are happy with your settings and preview it is time to export the sliced gcode file. Plug in an SD card (or memory stick when using the Ultimaker 3). Then click Save to Removable Drive to save your file. Make sure you take note of the file name so it's easy to find on the printer.

**Print**

To open the doors of the print cabinet please press the door release button, this will turn on the extraction and unlock the door after ten seconds. Please don't force the doors open. After the ten seconds the LED should turn green and the magnets on the doors will release. You will need to do this when starting a new print or removing a finished one.

Once the doors are open, 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**

If you are new to 3D printing please show your 3D model to a technician so they can check the geometry is suitable for printing.

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.

Please speak to a technician if you find the nozzle is blocked, the material has run out or you encounter any other technical problem.

If your print is estimated to be more than 100 grams or 12 hours, you will have to pay for any extra material. The price is 50p per 10 extra grams of PLA.

Please get a quote from a technician and pay online via: [www.instituteofmaking.org.uk/shop](http://www.instituteofmaking.org.uk/shop)

If a print you have paid for fails, the first retry is free.

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!