UNIVERSITY OF CAMBRIDGE
Department of Engineering

Dyson Centre
For Engineering Design

User manual for
Ultimaker 3 Extended 3D printer

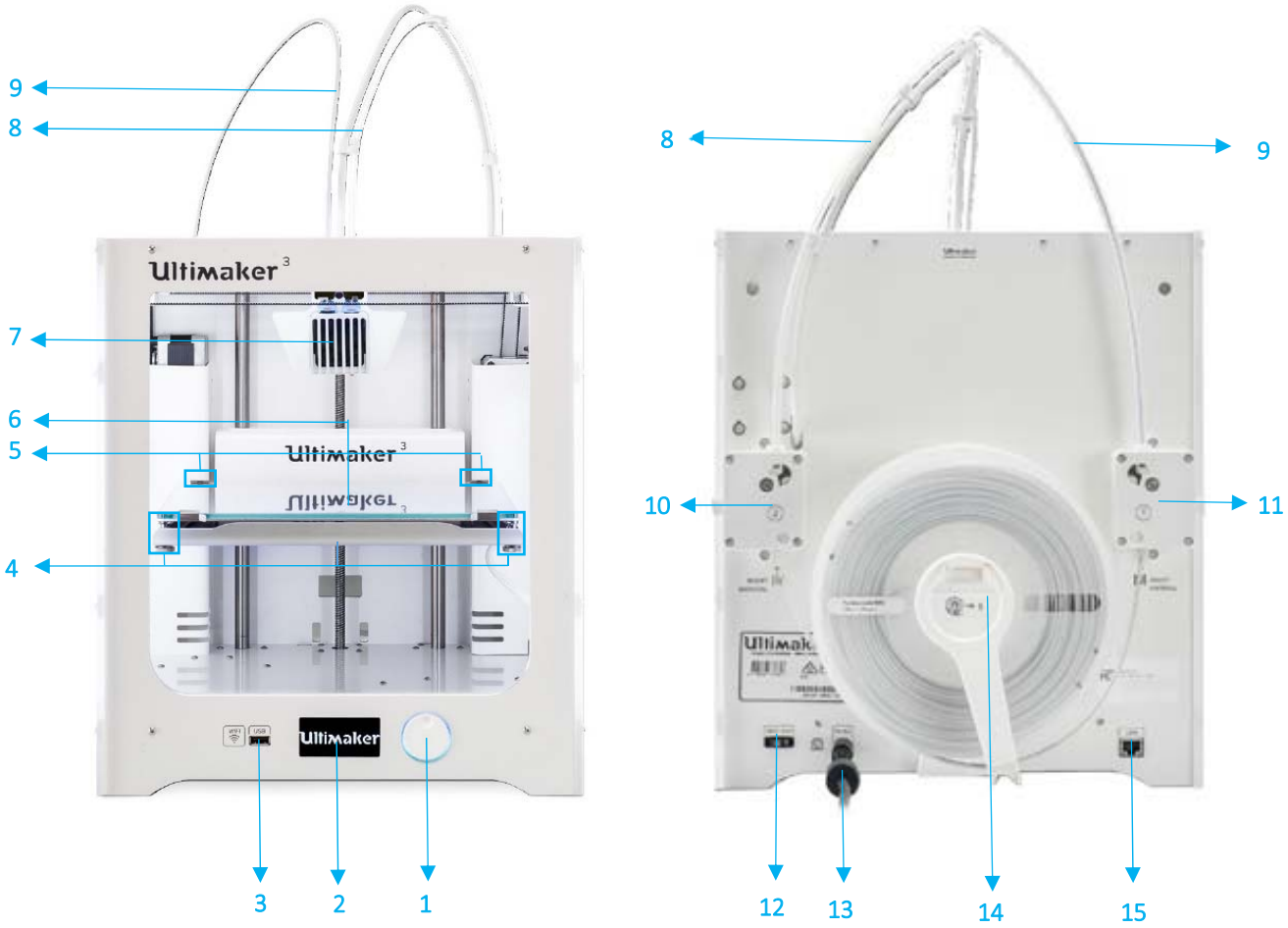
Created/edited by	Version	Date
Diana Thomas-McEwen	V1	12/09/2017

Table of Contents

Ultimaker 3 Extended 3D printer User Manual

1.0 Machine Familiarisation

1.1 Basic machine exterior set up

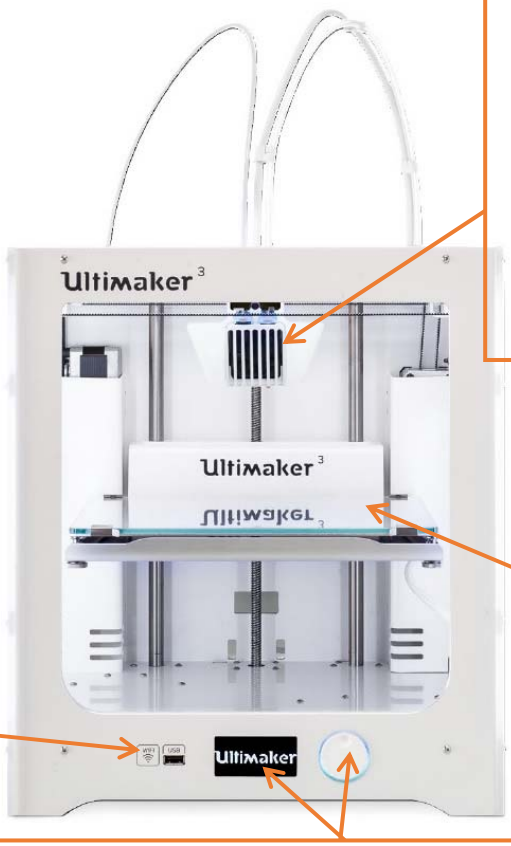


1	Push/rotate button	5	Build plate clamps	9	Bowden tube	13	Power socket
2	Display	6	Build plate	10	Feeder 1	14	Spool holder
3	USB slot	7	Print head	11	Feeder 2	15	LAN socket
4	Build plate screws	8	Print head cable	12	Power switch		

1.2 How do they work?

Extruder Head

- Moves side-to-side, front and back
- Mechanism to draw filament in.
- Then heats filament above melting point
- Small nozzle out of which molten plastic is forced
- The two separate lights indicate the individual nozzles.



Buildplate

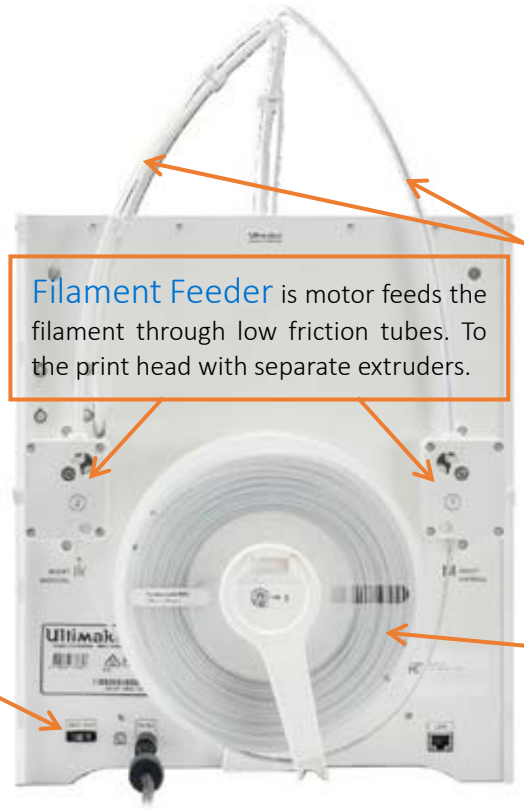
- Moves up-and-down
- Can be heated
- Removable glass sheet is clipped to the platform (this is what is printed is on)

USB slot

- USBs are provided for using the printers
- Used for file transferring to printer

Push/rotate Button & LED display menus

for operating the printer, setting up printing and use the rotate button to select.



Filament Feeder

is motor feeds the filament through low friction tubes. To the print head with separate extruders.

Filament

is drawn through low friction tube into the extruder head

Switch

on and off on the rear

Reel

of 2.85mm diam. Filament spools off the duel reel.

The extruders can have temperature set to anywhere between 180°C and 260°C and the platform can have temperature set to anywhere between room temperature and 100°C. The ability to set both temperatures governs the ability to print different materials. The extruders must be able to melt the material to be printed and heating the bed prevents shrinking when the print cools and warping off the bed.

WARNING: Caution also has to be given to any dangerous gases which might be given off on heating a material such as ABS– don't assume it is safe!!!

- PLA and PVA do not known to give off these gases.

2.0 Materials

There are a variety of different materials that this 3D printer can be use due to its range in temperatures.

2.1 PLA

PLA (polyester called polylactic acid or polylactide) is derived from renewable sources such as corn starch. This makes it biodegradable and recyclable therefore easier to dispose of and with no harmful gases given off when used. The extruder melts the PLA at a temperature of 220°C, by heating the bed to a temperature of 70°C for print of anything of significant size or height can maintain adhesion to the bed. The Dyson Centre currently stocks natural (off white), red, blue, orange, yellow, black and grey at a price of around £25 per kg (for reasonable quality filament). Other colours can be ordered such as white, clear, grey, green, purple. Also (for up to twice the cost) fluorescent colours, gold colour, translucent filament, thermochromatic and photochromatic colours wood colours, filaments with sparkly inclusions.

"PLA is an organic material, often made from corn starch or sugar cane, making it quite safe to use as far as fumes are concerned. Currently there is very limited official information available about the effects of PLA on our health." Taken from http://3dprinthq.com/desktop-3d-printer-safety/

2.2 PVA

PVA (polyvinyl alcohol) is a water-soluble support material suitable for multi-extrusion 3D printing. With a good thermal stability, Ultimaker PVA is ideal for printing complex models that require supports for large overhangs, deep internal cavities, and intricate geometries. Designed for a seamless 3D printing experience, PVA provides good adhesion to PLA, Tough PLA, CPE, and Nylon. The extruder melts the PVA at a temperature of 220°C, by heating the bed to a temperature of 60°C for print of anything of significant size or height can maintain adhesion to the bed. The Dyson Centre currently stocks natural (off white) at a price of around £45 per 500g.

2.3 ABS

ABS (acrylonitrile butadiene styrene) is made from fossil fuels and therefore will release harmful gasses and like PLA, is recyclable. There are some biodegradable versions now coming on market, but most are not biodegradable. The extruder temperature needs to be at 240°C with the bed temperature at 100°C or higher to create a successful print, the area also need to be ventilated due to the gasses released. Suppliers stock a range of colours however, due to the problem of ventilation the Dyson centre does not currently use PLA. ABA is technically stronger than PLA, however doesn't bond to itself as well, resulting in 3D printed parts are about same strength as PLA. ABS is also a little bit more temperature resistance than PLA.

“A few studies have shown ABS fumes to be toxic to rats and mice and there is a fair chance that ABS fumes are more harmful than PLA fumes, mostly due to higher levels of emissions and higher toxicity.” “Recent studies have shown that ABS emits around 10 times as many UFPs (Ultra Fine Particles) as PLA when heated, again making ABS more likely to affect our health than PLA.” Taken from http://3dprinthq.com/desktop-3d-printer-safety/

2.3 Other material available

Many other options are available including:

- Metal laden filaments
- Wood laden filament (sometimes referred to as wood)
- Flexible filaments (Flexi-PLA or Ninja flex)
- HIPS (High Impact Polystyrene) also dissolvable so might be useful for making other mould, which can later be dissolved away.
- PC (polycarbonate)
- PA (polyamide)
- Nylon

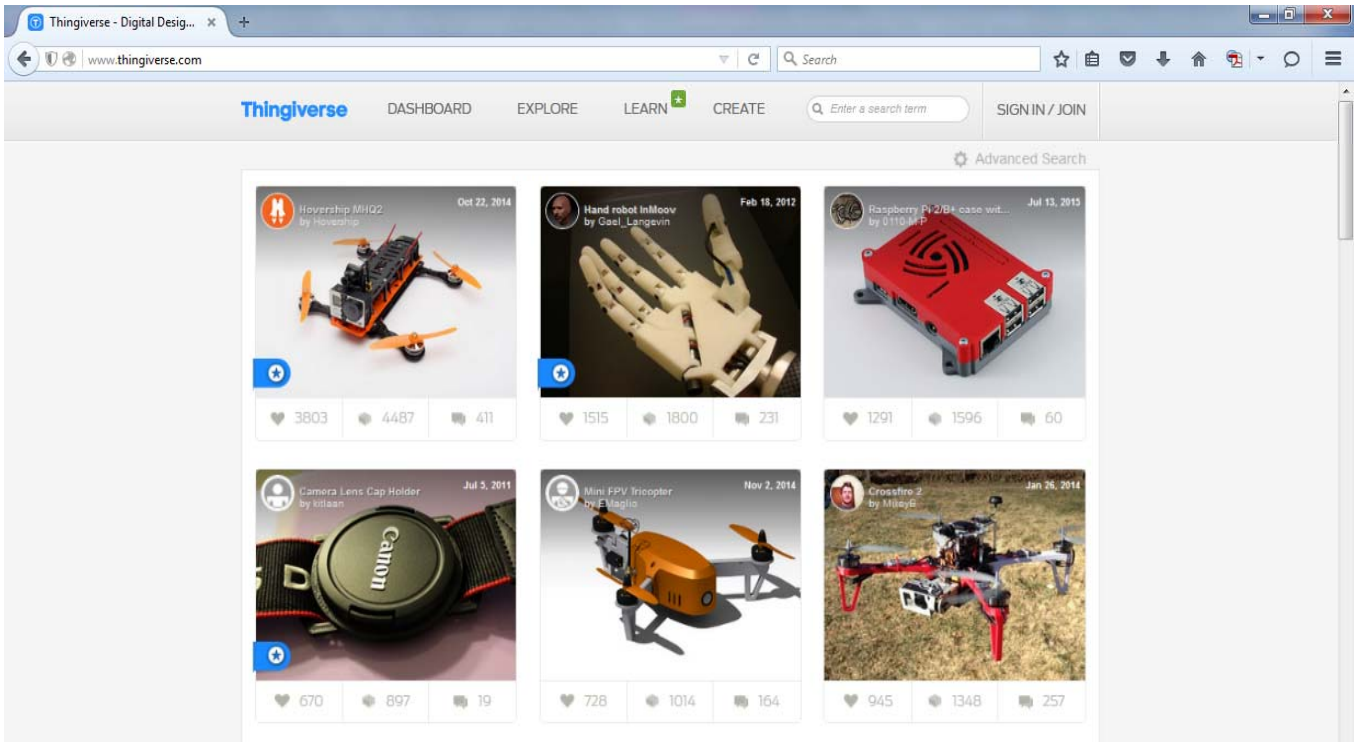
We have yet to investigate any of the above, in terms of availability (in 2.85mm diameter filament) / cost, Safety of fumes and how well our printers work with them. The printer may be harmed by using them (e.g. acidic compounds given off on heating) so further investigation is needed before using these in every-day use.

3.0 Where can I get the design?

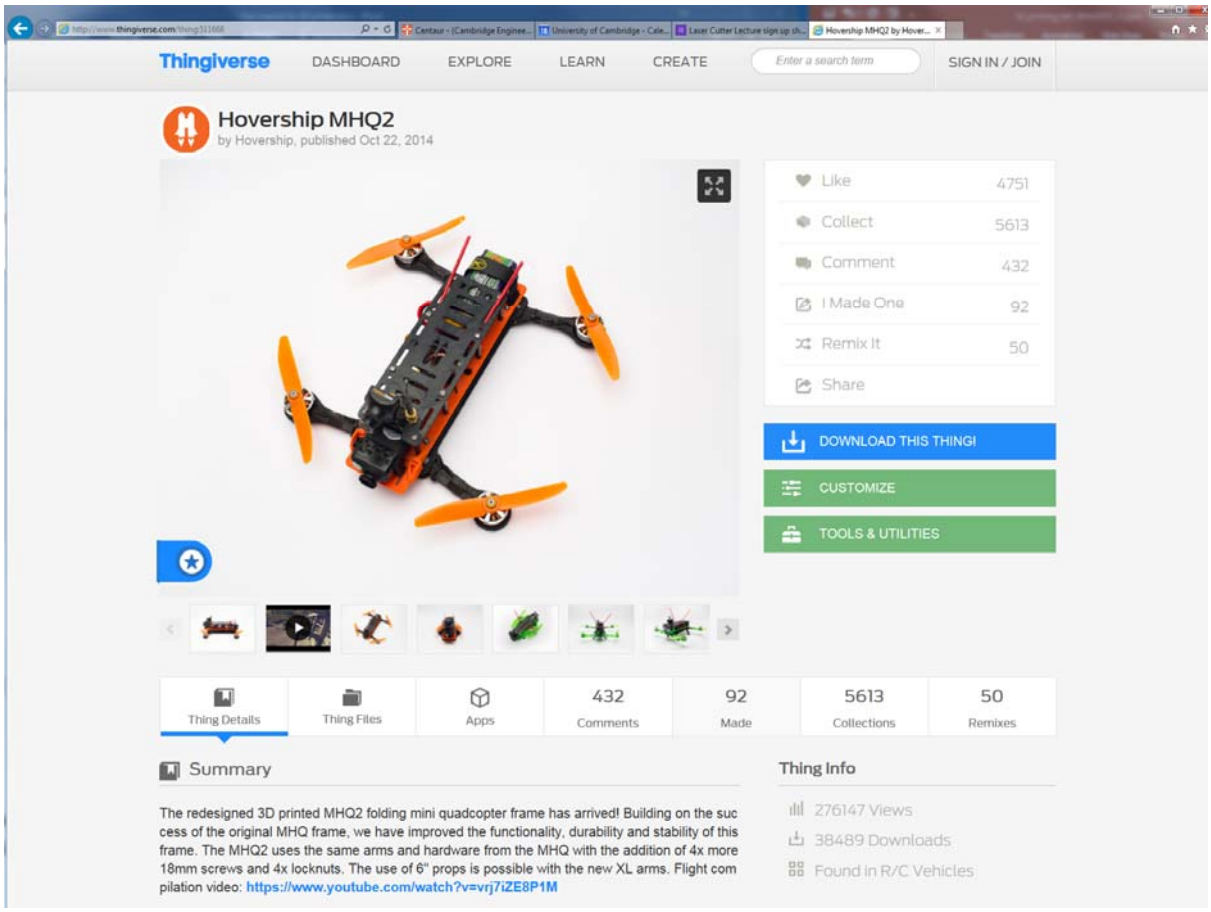
There are many online databases that offer free downloadable designs that just need to be converted into the right file format.

3.1 Obtaining an STL File

Websites such as Thingiverse (a universe of things!) and instructables makes this very quick/easy to obtain STL: Standard Triangle/Tessellation Language (or other similar file formats



Simply go to the website type in the item or theme that you are interested in downloading, search through of thousands listed designs and download the files needed to create it.



3.2 Designing a file

The STL file generates very sensitive settings so the user will need to check the STL file provides closed (watertight) surfaces. Still worth checking even in the case of Thingiverse STL files, or if file has been 3D printed before as different printer software processes STL files differently.

If drawing in any of the 3D drawing packages below please see, www.dysoncentre.eng.cam.ac.uk/stl for help outputting STL files.

- [formerly Google's] Sketchup
- Creo (Pro-Engineer)
- Solidworks
- AutoDesk Inventor
- Catia
- IronCAD
- McNeel Rhino
- Solid Edge
- UGS NX

Note that molten plastic has to be supported, so consider adding supports in your model (please see section--- for further details).

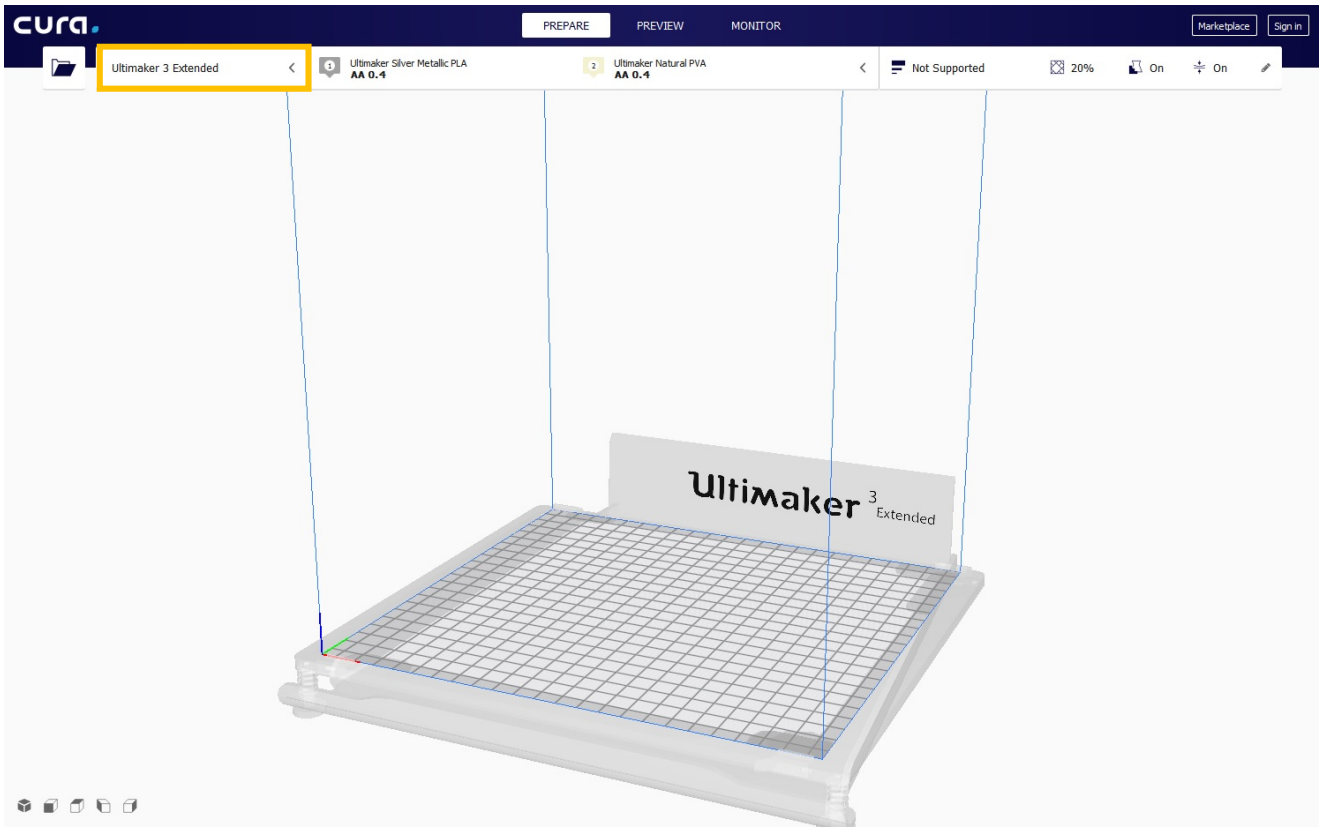
Inspiration of what to print and the uses for 3D printing designs can also be found on the Ultimaker website along with advise on using the Ultimaker software, Cura and technics in how to get a successful print.

3.3 Processing the STL file using 

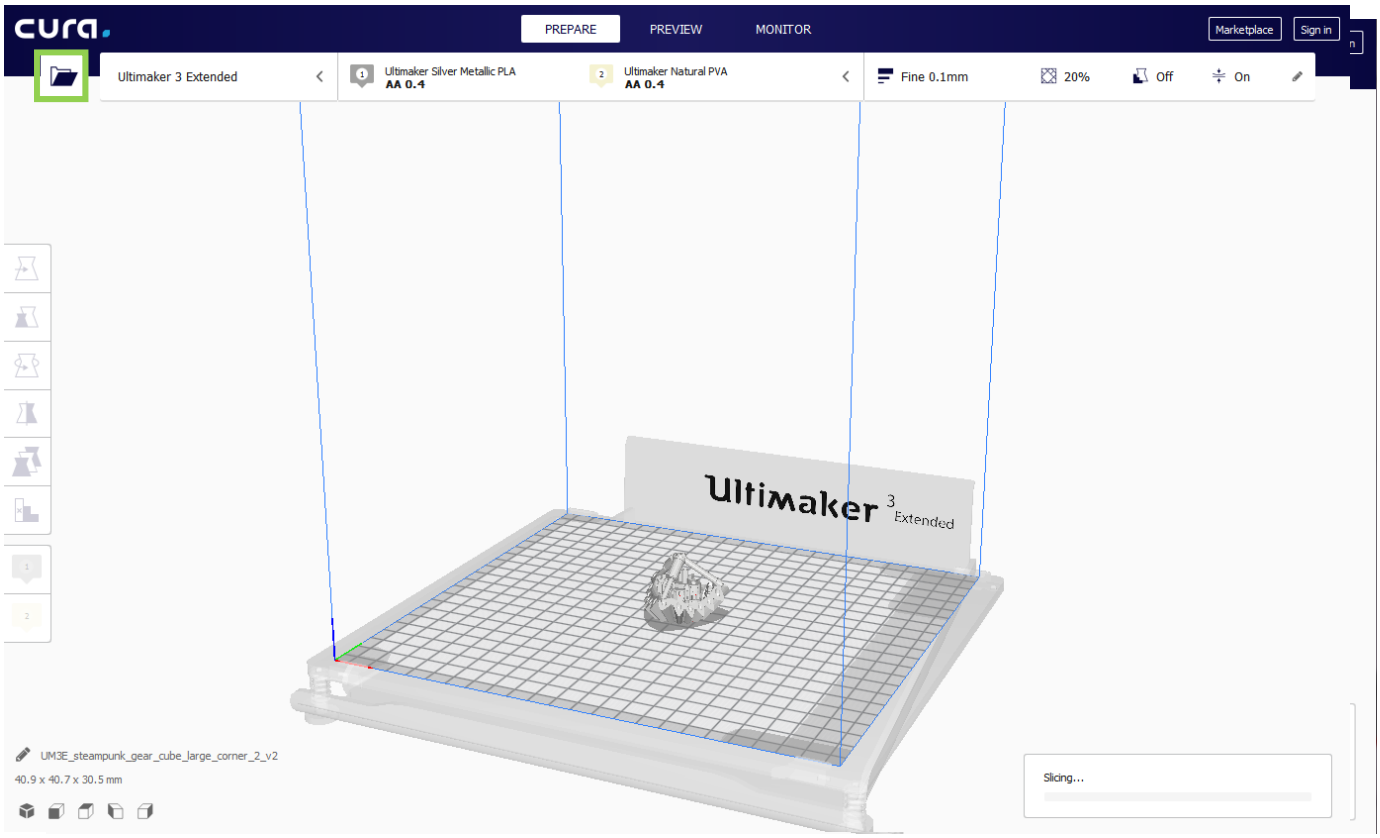
Ultimaker Cura 4.0.0.

To process the STL file to create the GCode file which can be fed into the Ultimaker printer, we use a package called Ultimaker Cura 4.0.0. You can install this on your own PC via the Ultimaker website (https://ultimaker.com/en/products/cura-software) or it is installed on the machine/s indicated near the 3D printers.

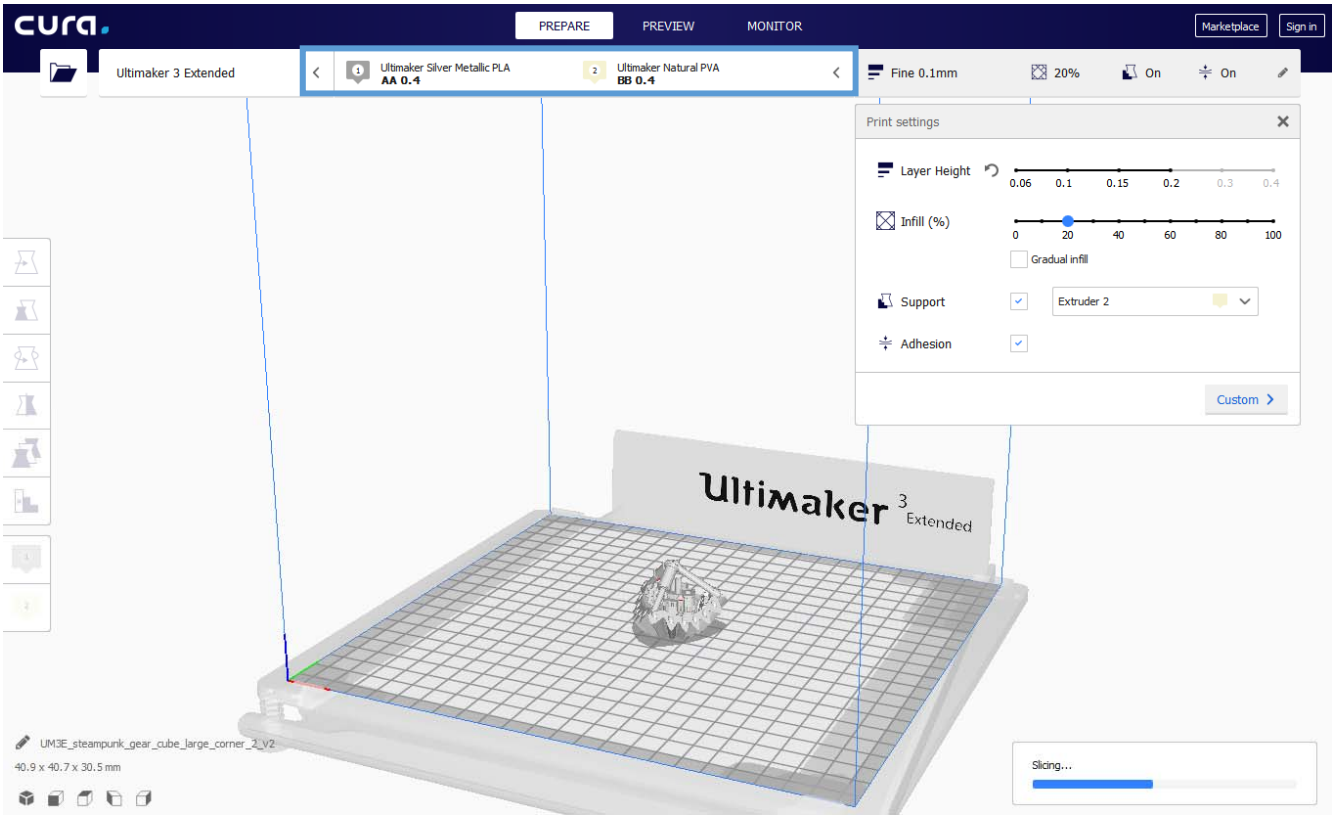
The software takes the arrangement of triangles in 3D space, which represent your surface/s and turns them into instructions for making 2D layers. Instructions for such machines (including CNC milling machines) is known as GCode, which is instructions such as move X by so many millimetres, or move in an arc in the XY plane, centred on this point, etc. The created GCode can then be used to print on the Ultimaker.



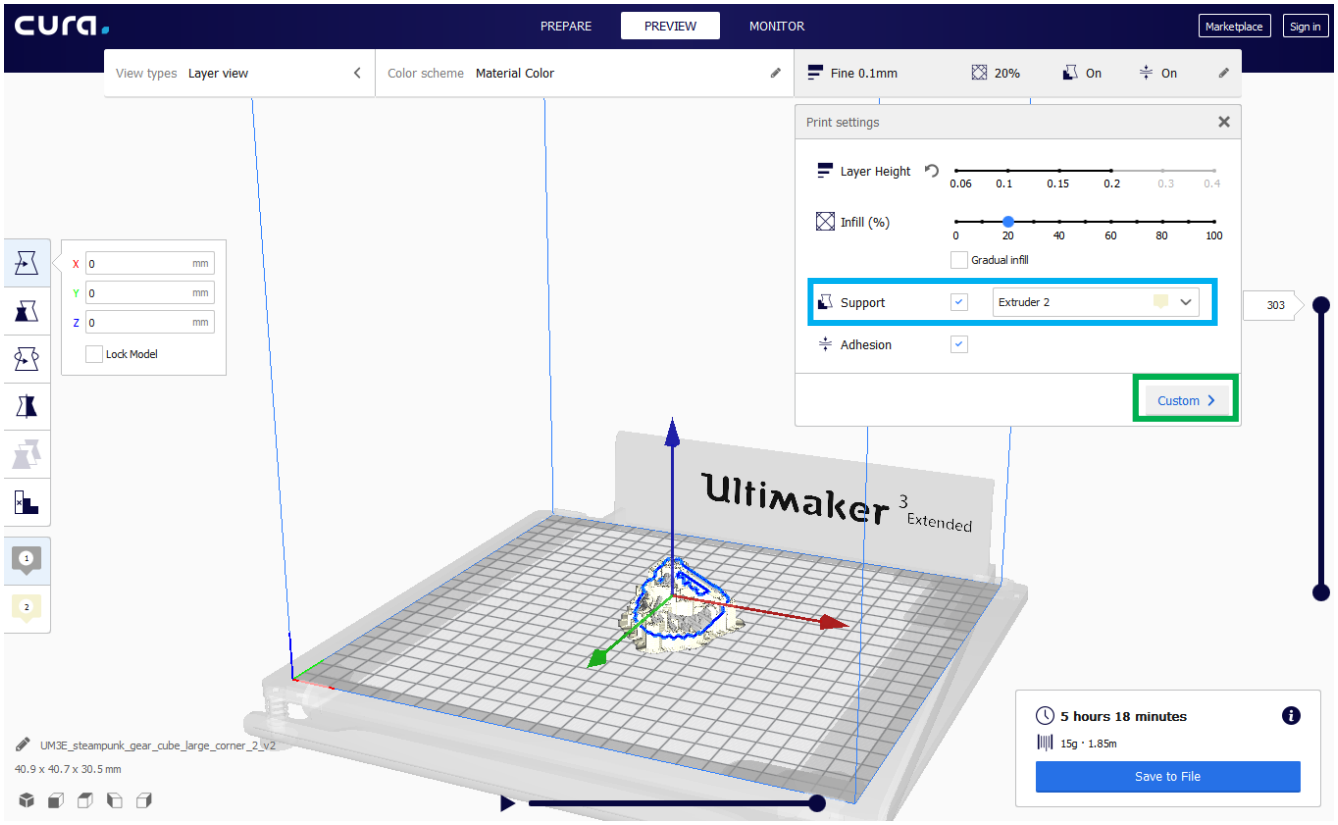
1. Open the software and select ‘Ultimaker 3 Extended’ as the printer (may need to go to ‘settings’- ‘add printer’).



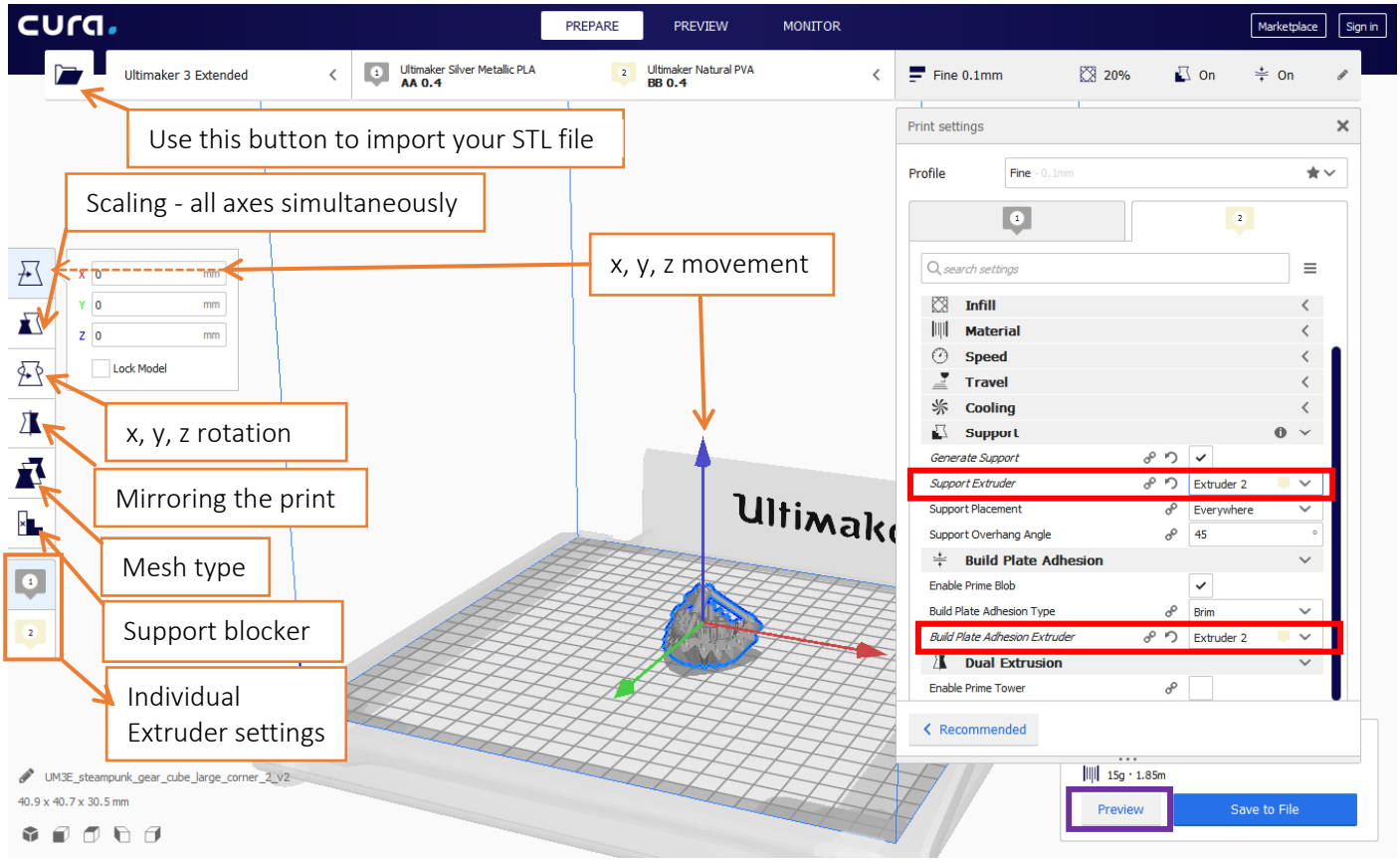
2. Open the STL to be printed and allow the software to slice the design.



3. Select the material for the two different nozzels and ensure the first nozzle is set to ‘AA 0.4’ and the second nozzle is set to ‘BB 0.4’.



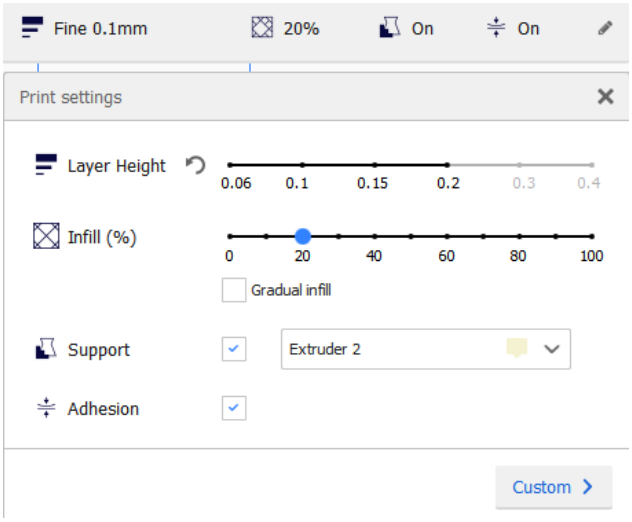
4. In the ‘Recommended options’ the layer height and ‘Percentage (%)’ infill can be adjusted here (unless necessary this should be left as default). Make sure ‘Support’ is ticked and is set to ‘Extruder 2’ for any over hangs and ‘Adhesion’ is ticked. To make more specific adjustment to the print setting select Custom (this view is of the preview screen, see next step for details).



5. In custom extra adjustment can be made to the composition of the print in this. Adjust the size, orientation and position on the bed. Ensure the support material is to be printed in PVA by changing the extruder from 'Extruder 1' to 'Extrude 2'. Selecting 'Preview' will show a preview of the finished print including supports.

Recommended options

These setting will be default depending on the materials and the printer you have selected.



Layer height

This is the height of each layer as it extrudes. Higher values will create a fast low-resolution print, whilst a low value will produces a slow print of high-resolution.

Infill

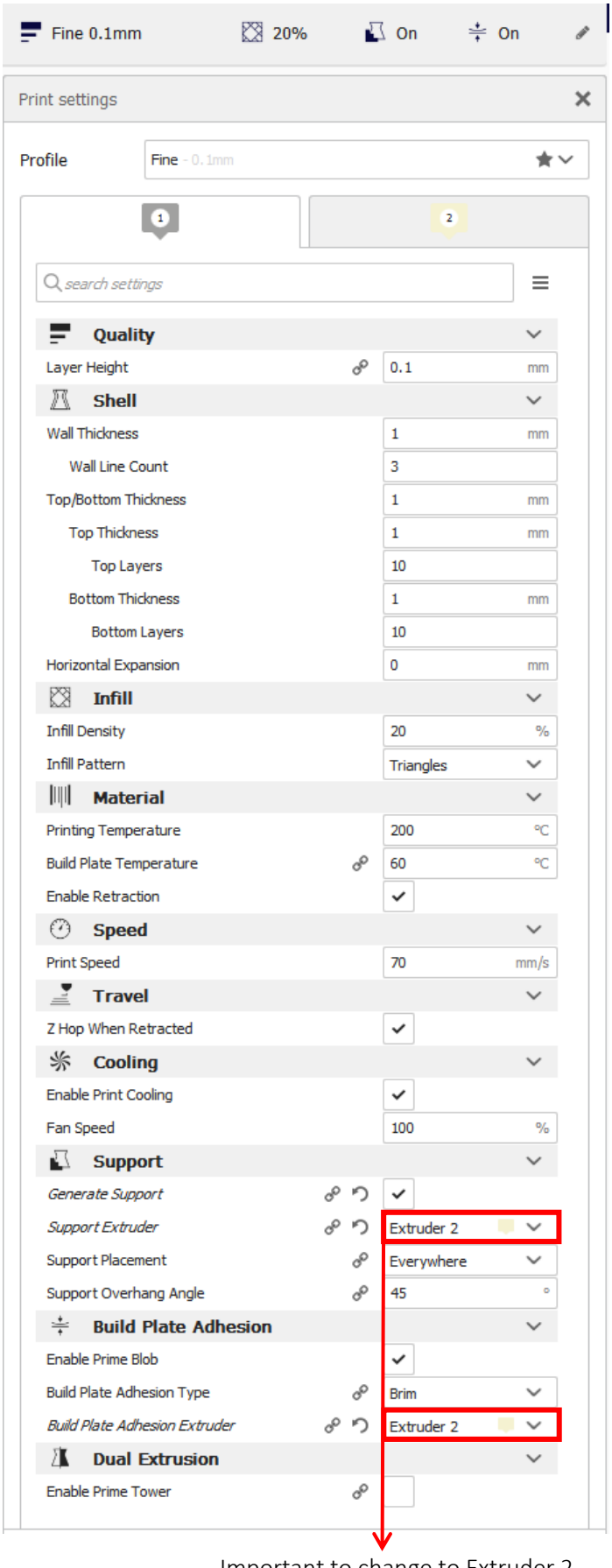
3D printed objects usually consist of a honeycomb centre and a solid shell the drop down for fill density allows you to vary the amount of material in the honeycomb in the form of a percentage.

Support & Adhesion

Support can be added to support any overhanging area, if using PVA support material to be set to extruder 2 will enable you to dissolve the supports after printing created a more polished print. Selecting Adhesion will add a brim or raft to enable.

Custom options

For more options select the Advanced setup tab. There are two separate tabs for each of the extruders to the setting for each of these can be set individually.



Important to change to Extruder 2

Quality

The layer height can be modified to be thicker so the print will print quicker. How thick a layer (in mm) the printer tries to lay down at once, and how fast it is traversing will have an effect in print quality.

Shell

Enables you to vary the thickness of the solid shell. What to pick will vary a little depending on what you want the part for, but the default setting for both is not a bad start point.

Infill

3D printed objects are rarely solid usually they consist of a honeycomb centre and a solid shell the drop down for fill density allows you to vary the amount of material in the honeycomb in the form of a percentage.

Material

Allowing you to select material specific setting. Either PVA or PLA for the Dyson centre machines unless started

Print speed

Standard print speed seems to produce good results, as do faster and thicker settings for crude items. Faster printing may mean that layer below doesn't have enough time to set before the next layer is put on, therefore the print is more likely to come away from the bed.

Cooling

Adjustments can be made to cooling and on what layer this cooling should be from.

Support

Support can be added to support any overhanging area, this allows you to select the areas and from what angle and density the support is needed.

If using PVA support material to be set to extruder 2 this will enable you to dissolve the supports after printing created a more polished print.

Platform Adhesion

Adhesion to the platform is important to consider depending on the print being created. There are Three different option 'Raft', 'Brim' and 'Skirt'. A 'Raft' produces a layer under your item, which helps it to stay stuck to the print bed, whilst the item is being printed. It will easily peel away from your item. A 'Brim' produces a single flat layer around the base of the print to aid in preventing warping. Lastly 'Skirt' produces a single line around the print but not connected to it.

If using PVA support material to be set to extruder 2 will enable you to dissolve the supports after printing created a more polished print.

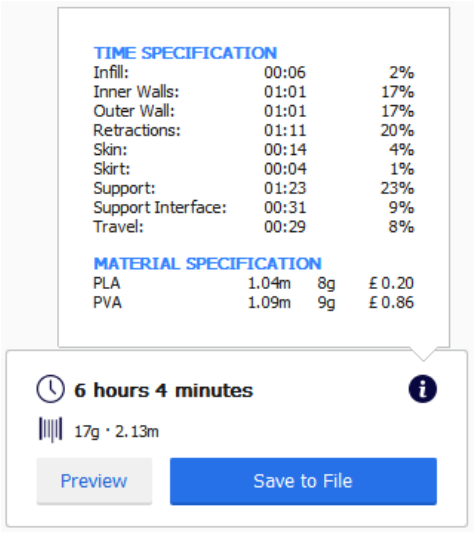
Duel Extrusion

With the 'Enable Prime Tower' ticked a tower will be printed next to the print which serves to prime the material after each nozzle switch.

NOTE: Temperature setting are set automatically

3.4.1 Saving the file to GCode file format to print

Click 'save to file' to generate the GCode, the progress bar will then start to show bar as it processes each lay of the design. Note that this is computationally fairly complex and that time taken may vary from a few minutes for something simple to about half an hour for the example item with no support to a few hours for a very complex piece. The rate that the progress bar moves is a useful indicator something is happening and how long it could take to complete.



Once the GCode has been produced ready to print take notes of the material specifications as the weight of the individual materials used (the PVA and the PLA). These are used to work out the cost for email of the materials.

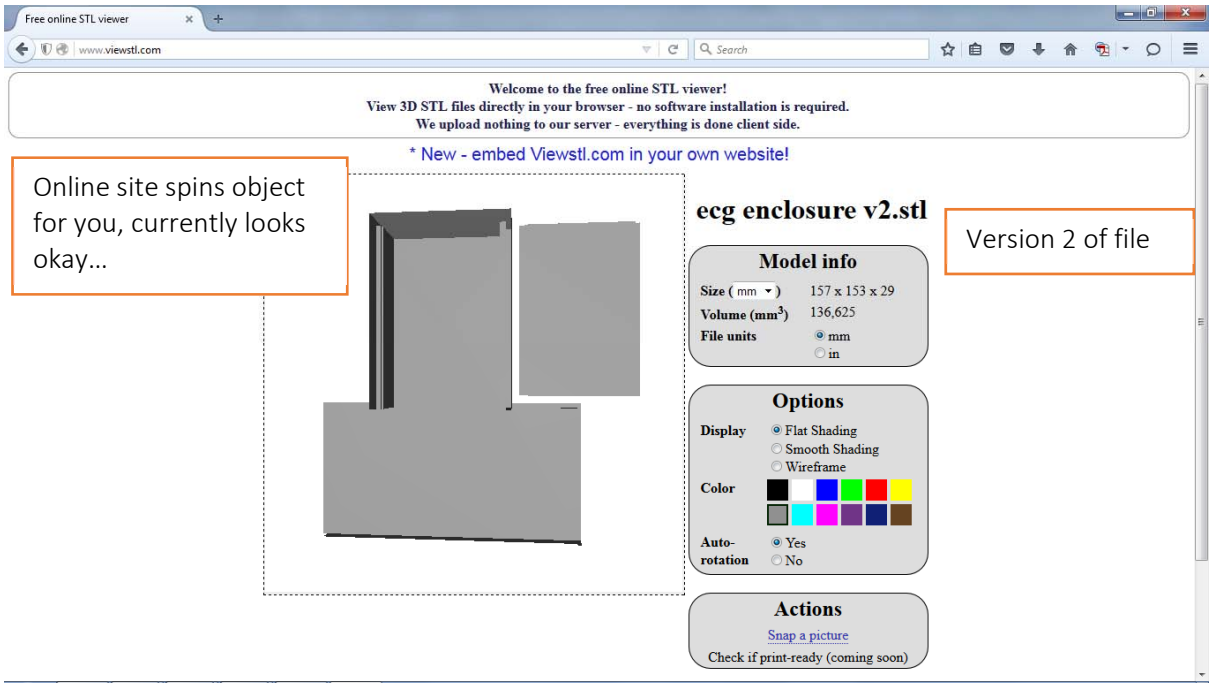
Take note of the number of grams (g) for each of the materials and write them on the user white board form in from of the Ultimaker 3.

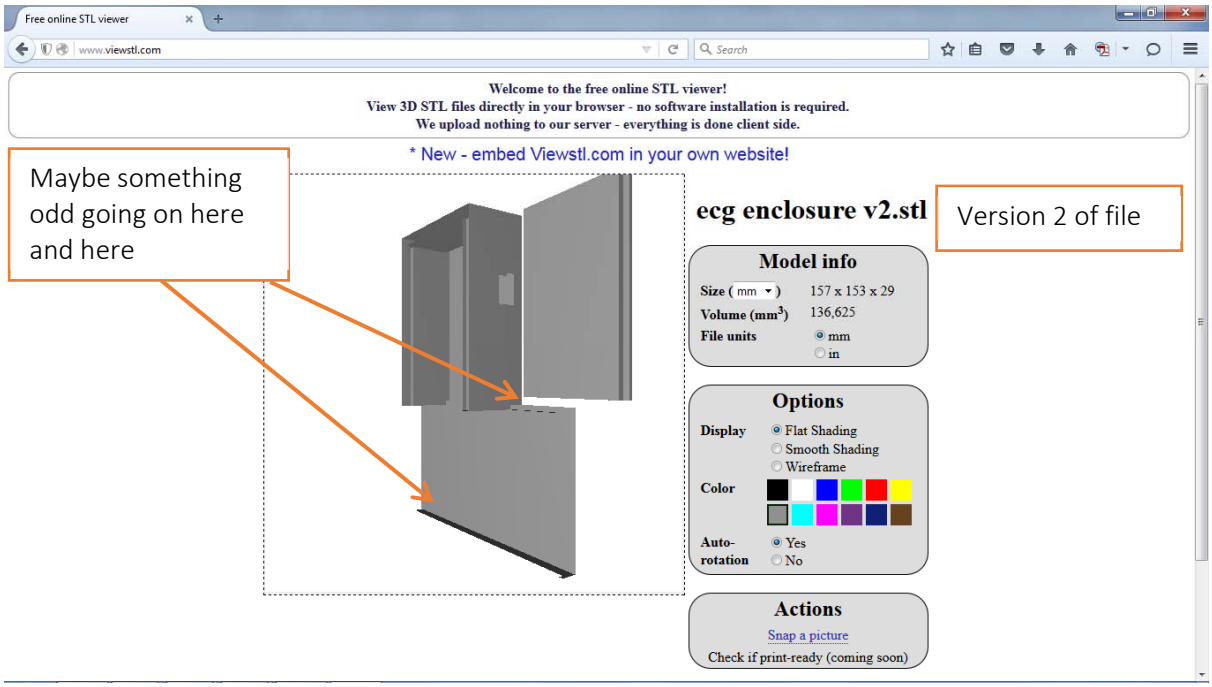
4.0 Trouble shooting - 'My file wont print properly'

If the print it missing or has areas filled in that were not included in the design then this likely to be a problem with the file so worth checking all parts during its processing.

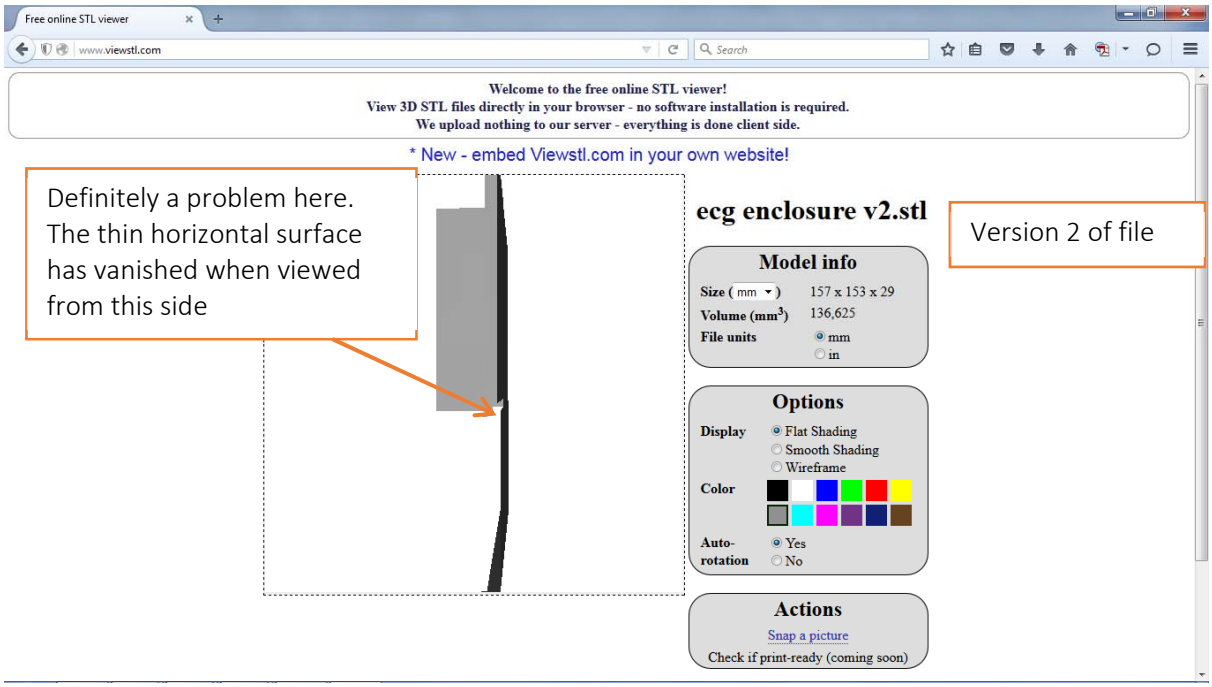
4.1 Checking the STL file for errors

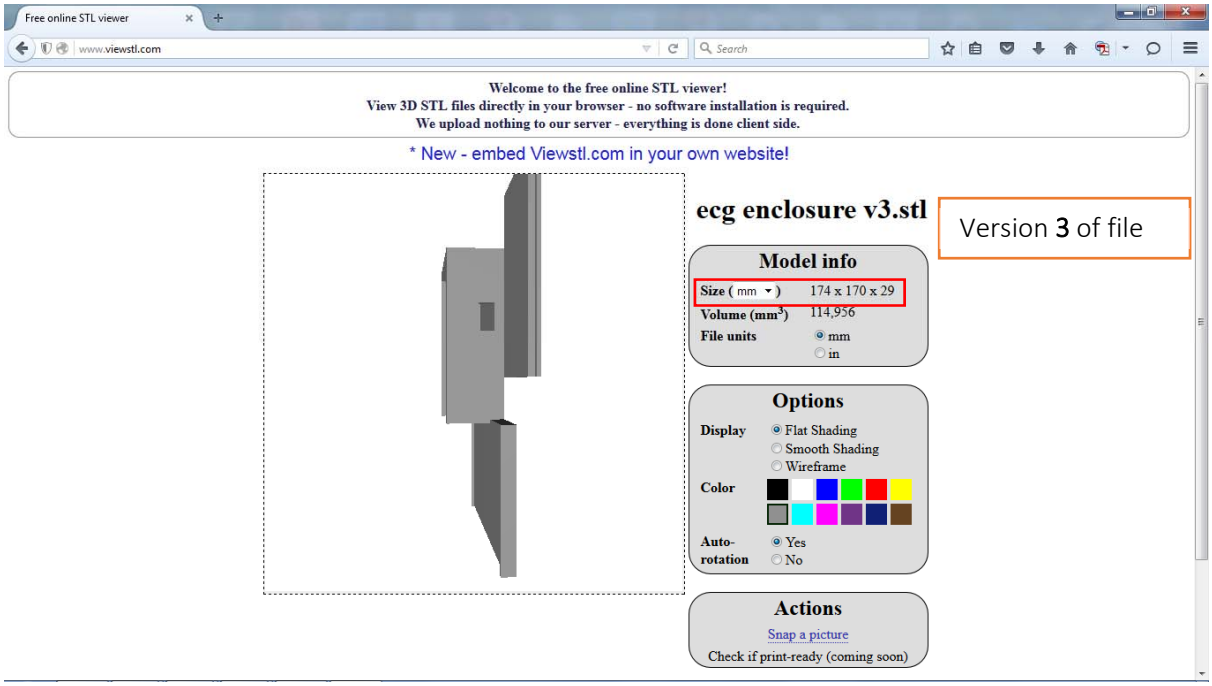
The STL is a mesh file created from triangles (Standard Triangle Language) used to map us the geometry of a 3D object form the original file. A quick check of STL files can be done with a viewer e.g. go to www.viewstl.com and drag/drop (or upload) your (<35MB) STL file.





This user then re-exported their STL file using different settings, and the problem seems fixed in this view.



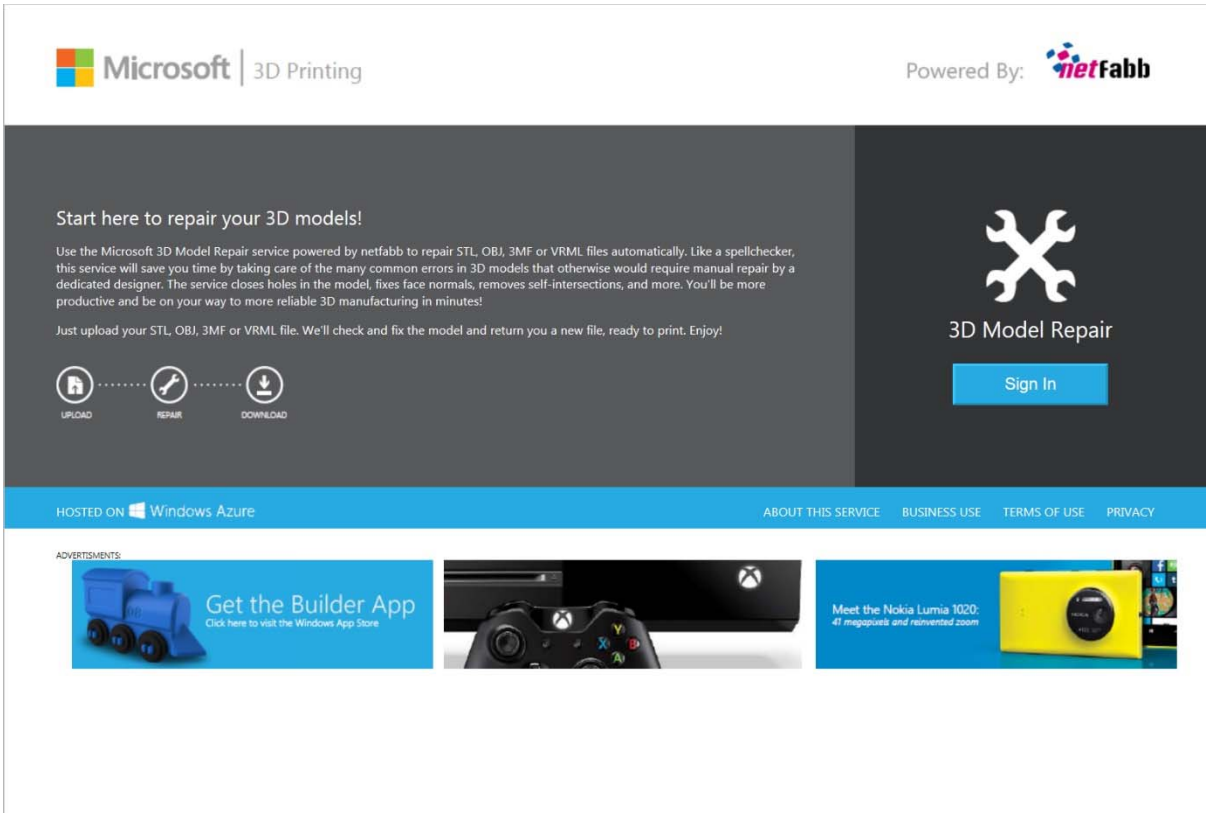


NOTE: It takes seconds to check in this way. The file won't print properly with 'issues' so tends to waste a few hours of printing time.

4.2 Fixing the STL file errors

Once the errors have been found in the STL file web sites such as the one below can be used to fix these errors.

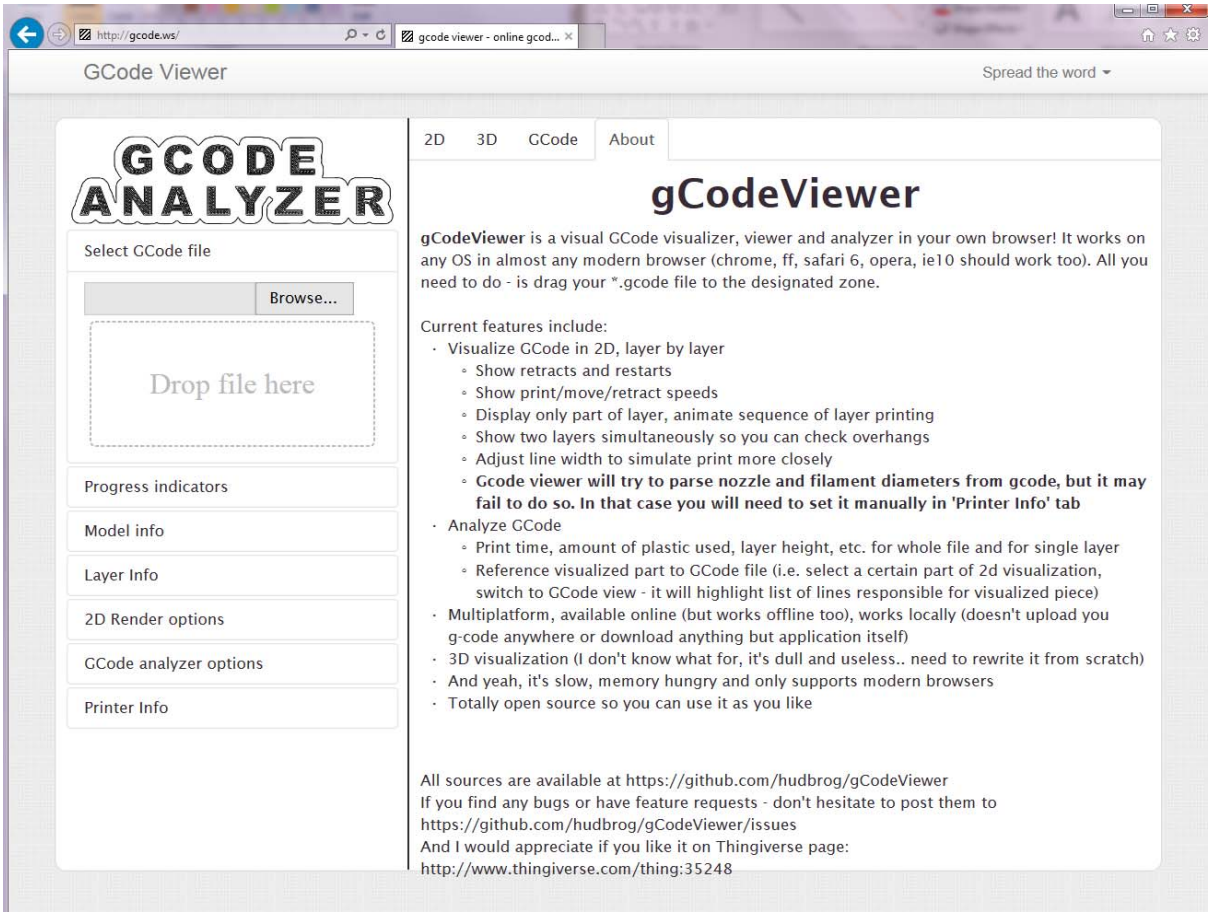
https://netfabb.azurewebsites.net/



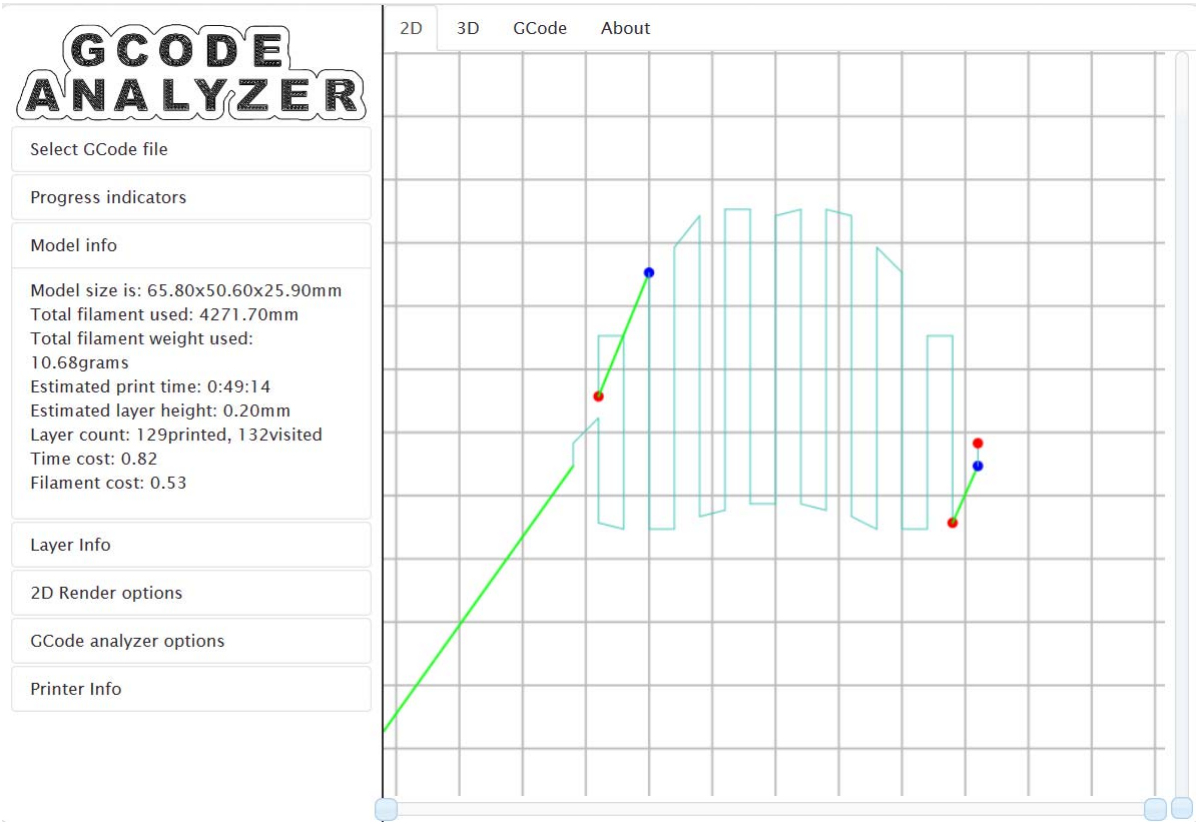
Sign in using a windows account (if you don't have an account you can create one here too) upload your STL file and click 'Repair' the site will go through your file and fix the errors, then you can down load the file. It is recommended that the file be viewed again in the STL checker. If errors still remain the file will need to be corrected in the program originally created in.

4.3 Tracking the problem with the GCode

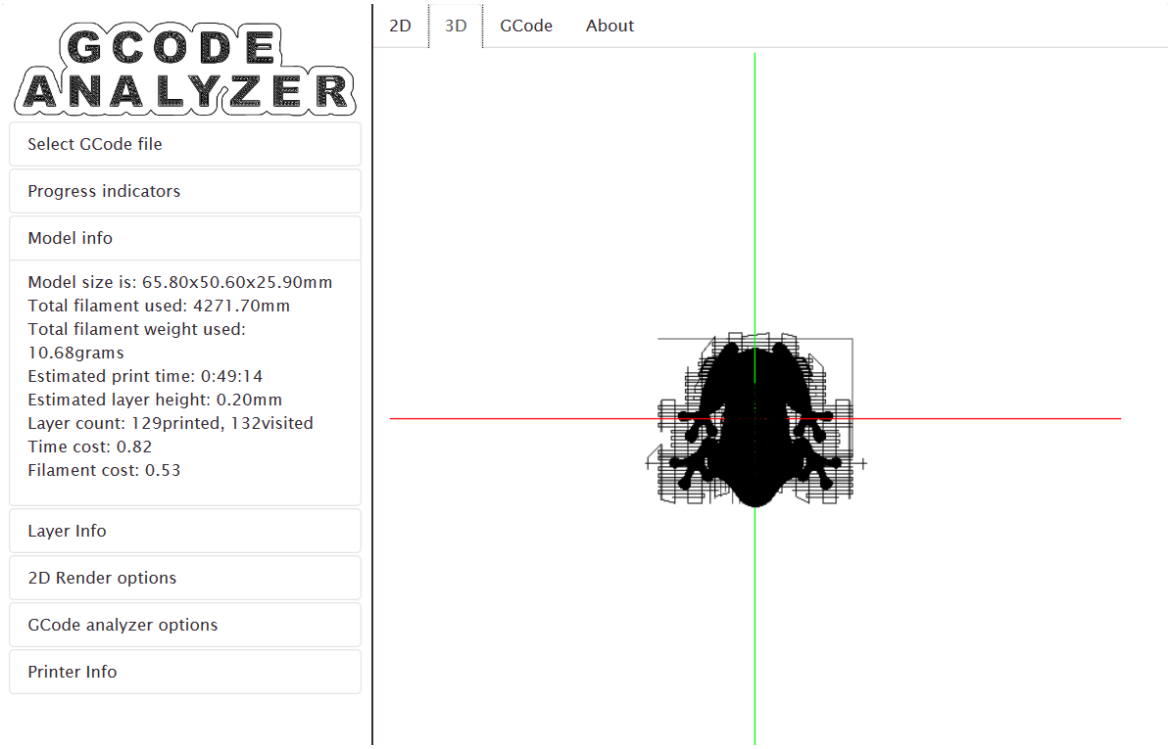
Sometime when producing the GCode, the print does not turn out the way you expect i.e. holes in gears are filled in. You can check that the problem is not with the GCode via GCODE ANALYZER, go to http://GCode.ws/



a. Upload your GCode

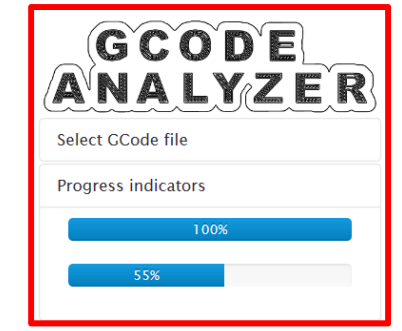


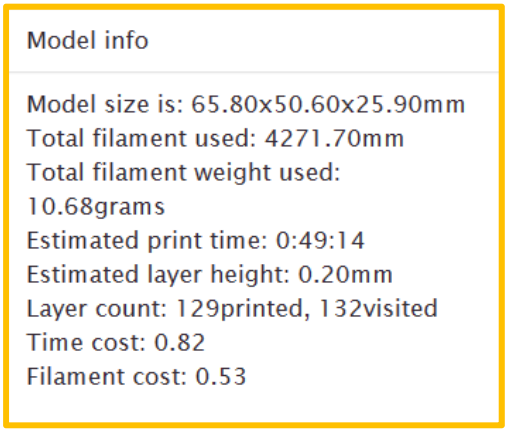
b. Looking at the 2D tab will show the root the raft will been created.

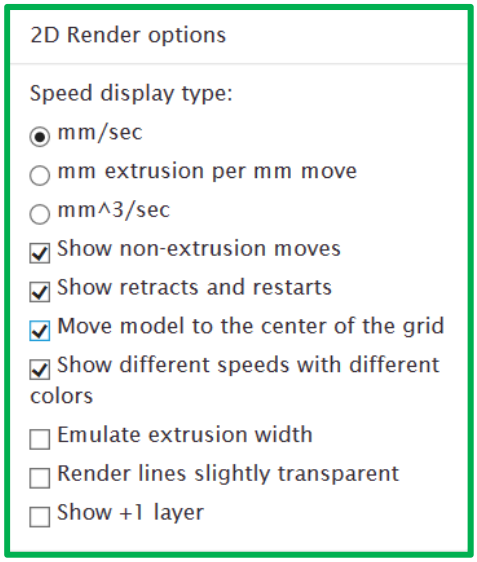


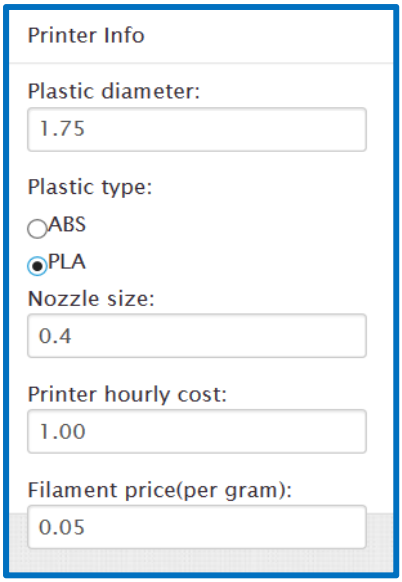
c. Looking at the 3D tab will show the model being created.

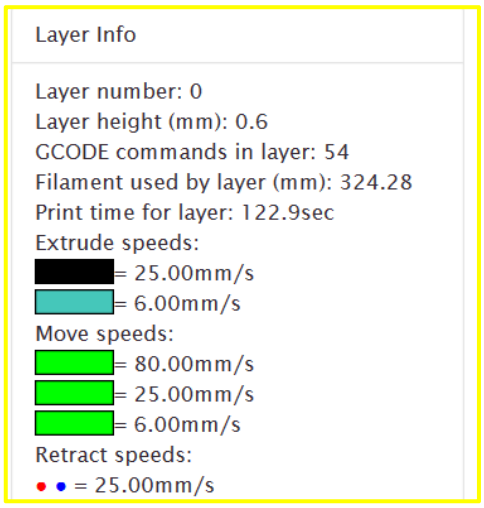
Other information included is Progress indicator, Model info, Layer Info, 2D Rendering options, GCode analyser options and Printer Info.

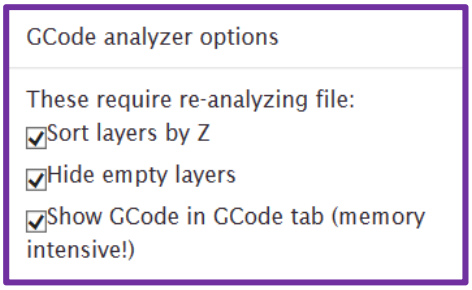


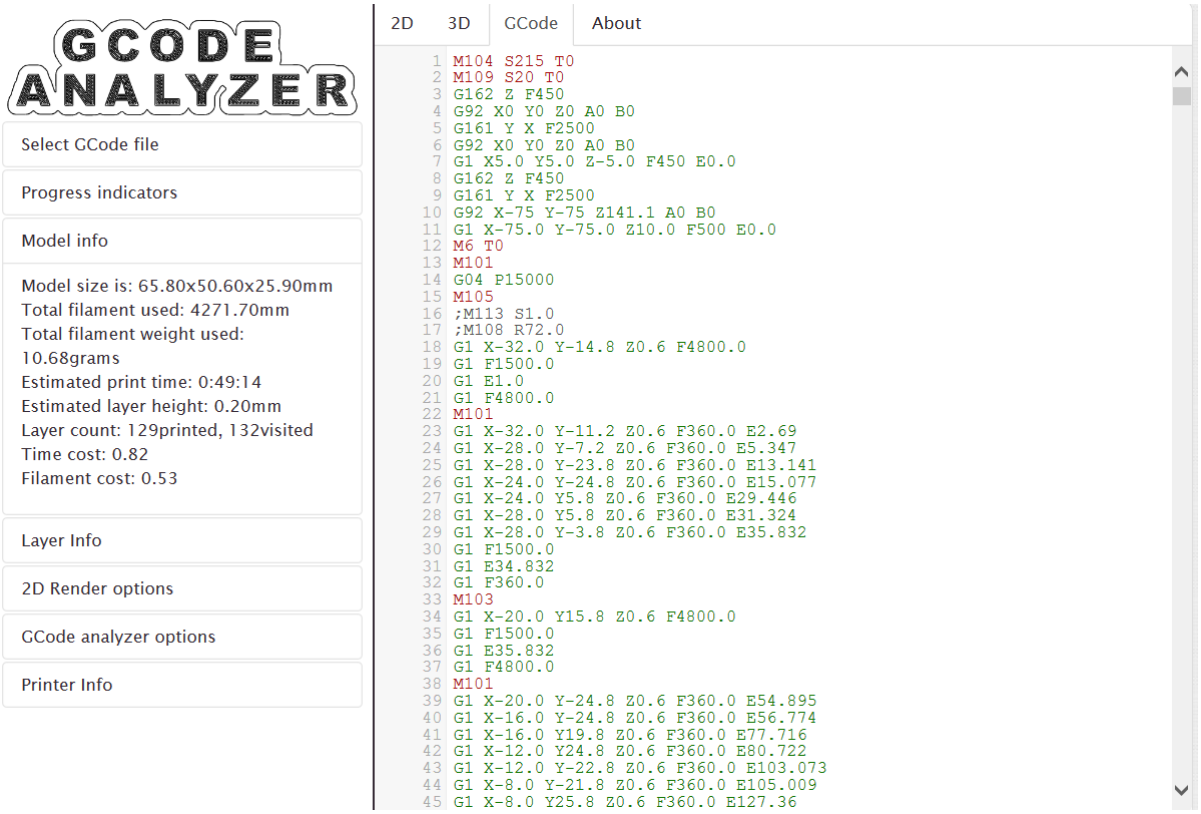












d. Looking at the GCode tab will show the code of the model being created.

5.0 Setting up the Ultimaker 3 Extended 3D printer

Once the GCode file has been created the printer can be set up, before using please check:

- The printer is on (at the wall and at back).
- Also that a glass bed is clipped with four clips to the aluminium platform.
- Check that the printer is loaded with the correct filament (both type of plastic and colour) located at the back of the printer and that there is enough filament to complete the print.

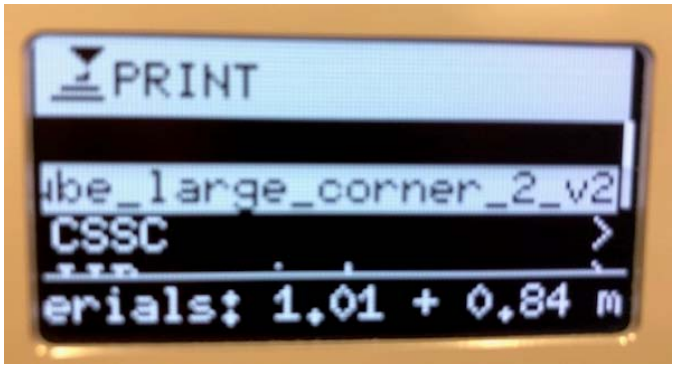
5.1 Setting up the printer and Starting to print

When the printer is first powered up the start-up screen will appear as shown below

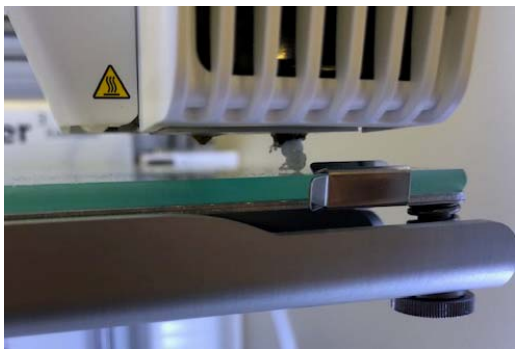
Insert your USB into the USB slot located to the left of the screen, now use the glowing dial to the right of the screen to select the 'Print' option (push the dial to select).

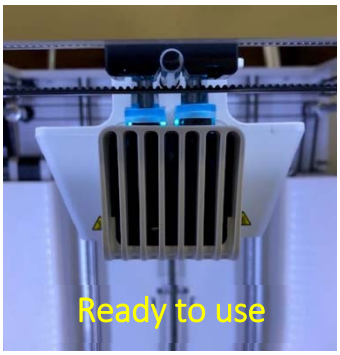
Until you insert your USB the printer will display this when print is selected.

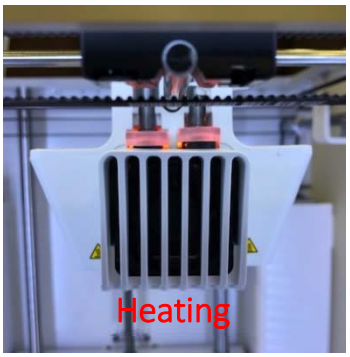
Select the file you want to print and push the dial to confirm the print file you have just added to the USB will display the most recently added file as 'Latest:…' on the screen, the name will scroll on the display once highlighted. Along the bottom information such as the print time and the material size scroll along for each highlighted print. Click the glowing dial to confirm print.

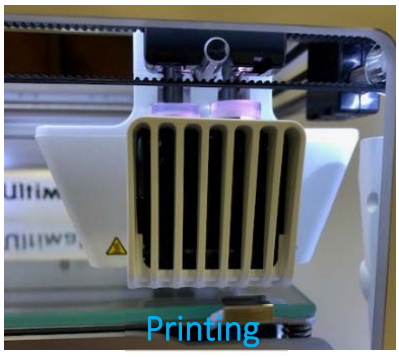


Once the print has been confirmed the printer will start to 'Prepare to print:…' by heating the build plate and then will flash red when heating up the nozzles.





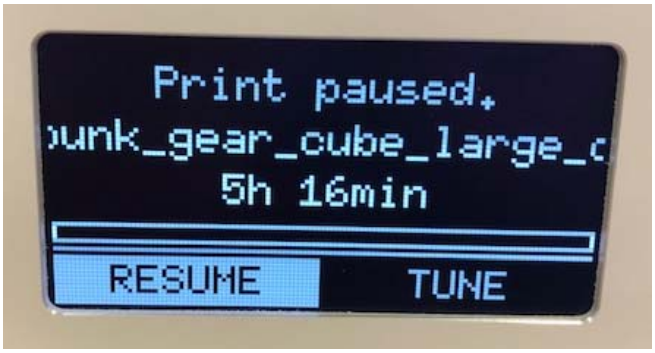




Now the print head will move into the right forward corner of the bed and position ready to auto calibrate the 'Z' and extrude a small amount of filament, before moving and doing the same on the left side. The printer will now start to print. The light on the head may also flash to show it is heating.

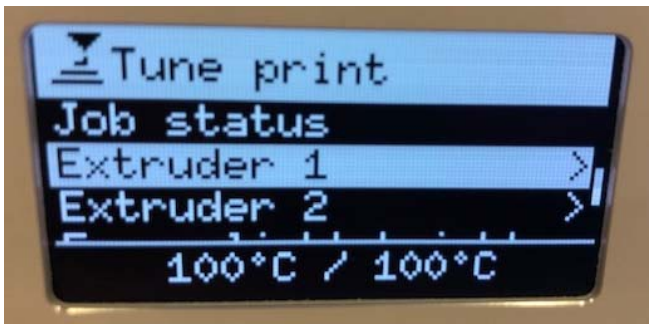
Once printing, the printer screen then displays two options 'Pause' and 'Tune'.

Selecting 'Pause' the head will pause extrusion and move to the right front corner and the plate will lower back the base. Select 'Resume' to carry on with the print.

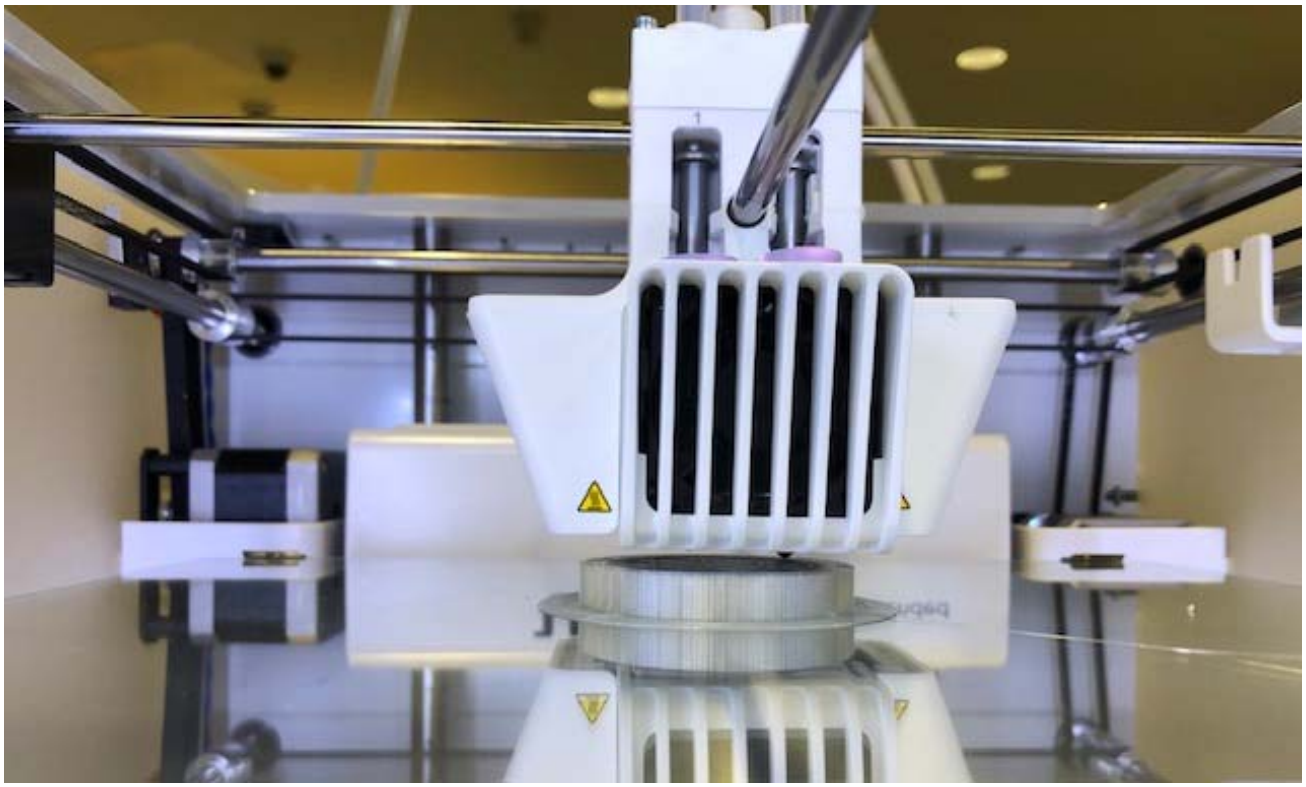


By selecting 'Tune' there is the option to aborting the print, when selected the printer will ask 'Your are about to abort this pin. Are you sure?'

There are other options in 'Tune' to adjust the printer settings such as print speed and build plate temperature and the individual extruder temperatures.



Each time the printer switches from one extruded to the other the printer will move to the back right corner away from the print (this would be where the 'Prime Tower' would be created if this had been ticked on the Cura settings)



NOTE: Occasionally the printer will perform a bed levelling calibration by moving to areas of the bed and lowering the nozzle to check the distance from the bed. This will not happen every print but once a day. If the bed is outside the calibration the printer will ask to be levelled before printing, please ask a technician to do this and do not attempt to do the calibration yourself.

5.2 Finished printing

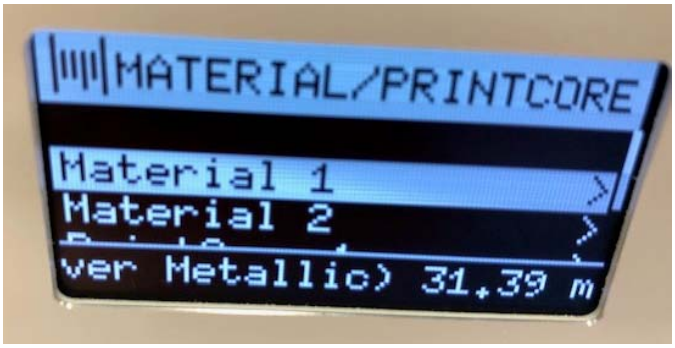
Once the print is complete it will drop the bed down to the lowest position and start cooling down, see what the current temperature of extruder and platform are.

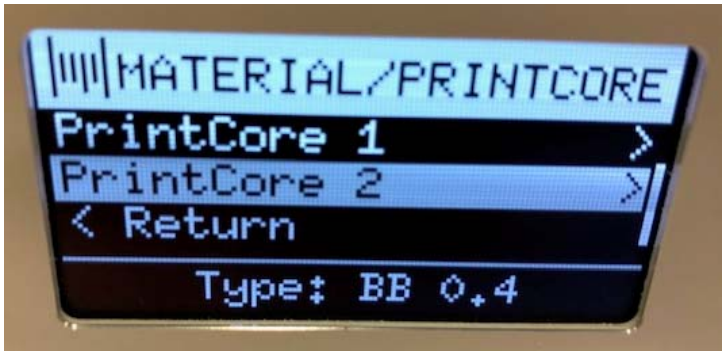
NOTE: Throughout the print the display shows its progress and how long is left, this is an estimation not an exact time, so please check on the print time throughout (if possible).

When you are happy the bed is cool enough (so as not to burn you, but also so that your component isn't still soft and pliable) use the spatula provided to push under the rim of the print to help remove the print from the glass, take care not to mark or crack the glass. If necessary pull/rotate the front two clips off the glass plate and slide the buildplate out from the printer to remove the print. Please then replace the plate and clips and don't forget to pay what is owed for the cost of materials.

5.3 Other Function

The other option on the start up screen should only be accessed by fully trained personnel.





The 'Materials' option will allow you to 'Change', 'Load'/'Unload' and 'move' the material and change the material type for each of the extruders. Selecting the 'printcore' allows you to 'Load'/'Unload the individual extruder heads and the setting (such as temperature) for each of the two extruder heads.

The 'System' also allows you to move and heat the head (as a whole) and the buildplate along with setting up a network connection (if the printer had access to the internet) and changing the language.

6.0 Charge forms

We would suggest that a print is trialled on the RS Ideawerk printers before using the Ultimakers incase the print is not what you wanted or how likely the print is to fail.

Only complex shapes that need disovlable internal support will be allowed to be printed on the Ultimaker 3.

We have funds to buy 3D printers for undergraduate use but we do not have funds to buy materials, so we ask you to pay for these, but the cost is relatively low. We do not have funds to pay for 3D printing for research, except for 4th year projects (or have enough printers for this additional use). If you are a postgraduate/post-doc researcher or a member of staff reading this – please contact the Dyson Centre Manager to discuss further.

6.1 How does it work?

Please use the electronic scales to work out the mass of PLA used and PVA, include any scrap, rafts, supports and all failed attempts (these are at your cost). The charge is PLA: £1 per 20g
PLA: £1 per 10g
of part there-of, for example for PLA.
18g is £1
30g is £2
40g is £2
41g is £3

For the time being please see the Manager to make payments and ensure you get a receipt. We will try to streamline the payment process in the future.

NOTE: All 3D printing filament payments will go into a dedicated account intended for buying more filament

Please take this to the Dyson Managers desk and pay upon completion

Dyson Centre
For Engineering Design

"A modern workspace where engineering students can come together outside of the classroom to think, experiment, design, build and exchange ideas"

Dyson Centre 3D Printing Charging Sheet

Machine	Mass used, including wastage / support (grams)		Cost per gram (£/gram)	Total cost (£) Whole pounds only		Not an Undergrad ?
RS Ideawerk Pro/Plus (WT280A)	Use scales provided to weigh parts		£1 per 30grams, ie 0-30grams: £1 30-60grams: £2			+£7
Ultimaker 2 Extended +	Use scales provided to weigh parts		£1 per 20grams, ie 0-20grams: £1 20-40grams: £2			+£28
Ultimaker 3 Extended	CURA PVA (g) / CURA PLA (g)	Total weight (g)	PLA : £1 per 20grams PVA : £1 per 20grams	CURA PVA (£) / CURA PLA (£)	Total Cost (£)	+£28

☐	*EPOS (payment will be recorded and checked) **VAT at 20%** will be **charged on top** if you are not an undergraduate*	**Total: £** (Including VAT)
☐	**Job number (please enter):**	**Total: £**

Contact Information

Name:_____

Date:_____ **Email:**_____

Signature:_____

Scan here to access the Dyson Centre payment system (For EPOS payments only) or visit:
https://tinyurl.com/DysonEPOS

Cash Receipt For £_____

Item description/Project name: _____

Name:_____Date:____/____/____

Manager's Signature:_____

Please note that these 3D printers have been paid for with the intention they are for undergraduate use.
For use by others for research purposes, please speak with the Dyson Centre manager be fore using.
PLEASE TURN OVER FOR TERMS AND CONDITIONS

7.0 Health and Safety

As with many such machines in workshops, these machines are not default safe to use, you can cause harm to yourself and others and to the machine if you don't understand how to use it correctly. An appreciation of the risks and what to do about them is important

7.1 What could go wrong

Could get burnt by extruder or by molten plastic coming out of it – usually between 200 and 280°C (significantly hotter than boiling water).

- Don't touch extruder, or molten plastic coming out of extruder, even if it looks cool.
- Please wear protective gloves provided (to be sufficiently insulated).
- Make sure you know where nearest sink is and get burns under cold water promptly and keep them there for 10 to 20 minutes!

Could get burnt by the platform/bed or item recently printed: if the bed is still being heated, it could be around 100°C

- Don't touch bed until you are sure it has cooled down – touch screen will tell you current temperature (be careful not to get mixed up with target temperature or same for extruder/nozzle).
- Be cautious when moving to handle the bed or the item printed.

Could get trapped above or below the moving [heated] bed and/or the moving [heated] extruder head.

- Keep clear off all moving parts

Harmful gases coming from extruder from heating non-standard materials
Or from overheating standard materials.

- Double check target and actual temperature of extruder.
- Double check what material is being used.
- Ask for advice if printing a non-standard material.
- If you do think you have been exposed to potentially harmful gases, summon a first aider as per three slides ago.

As the extruder is above the auto-ignition temperature of things like paper, there is a chance of the machine easily being able to start a fire.

- Keep the area free from paper, and any other debris.
- If realistically possible, stay with the printer whilst in use, or monitor it using the webcams which will shortly be provided.
- If you do find a fire:
    1. immediately raise the alarm.
    2. if safe to do so, use a fire extinguisher to try to put the fire out, and if possible disconnect the printer from the mains (assuming the electricity cables aren't now damp)

3. if in doubt, leave the building and raise the alarm.
4. be careful of breathing in any fumes from the fire.

Also seek help from a first aider – ask at reception if you can't find one, or if outside normal hours, call security on 31818 from the phone on the Manager's desk, or 01223 331818 on your mobile phone. Further details at Manager's desk.

Appendix A -Trouble shooting

Problem: My Print keeps coming away from the bed.
Solution: This could be one of four problems

1. The bed is not heating up
   - Check the temperature setting for the bed
   - Check the current setting to ensure it is heating
2. The extruder head is too far away from the bed
   - Ask a technician to adjust the Z value for the extruder head
3. The design being printed is at an angle that requires additional supports to complete the printing.
4. The bed is not level (please report this to a technician to be fixed).
5. The ambient temperature is too cold.
6. Brim it turned off
7. The print speed is too high

Problem: My design won't print; it heats up but then says that my print is complete
Solution: The GCode file might have been interrupted or did not fully save on to the USB, redo the GCode and resave the file on to USB. Also worth when producing the GCode, making sure it is produced on to the computer drive not directly onto the USB as that can sometimes corrupt the file too.

Problem: I find my file on the USB when I plug it in to print.
Solution: The file is not in the right format.

Appendix B - Ultimaker 3 Extended Specifications

Ultimaker 3 specifications

Printer and printing properties	Technology	Fused filament fabrication (FFF)
	Print head	Dual extrusion print head with an auto-nozzle lifting system and swappable print cores
	Build volume	XYZ: 215 x 215 x 200 mm (left or right nozzle only) XYZ: 197 x 215 x 200 mm (dual extrusion)
	Filament diameter	2.85 mm
	Layer resolution	0.25 mm nozzle: 150 - 60 micron 0.4 mm nozzle: 200 - 20 micron 0.8 mm nozzle: 600 - 20 micron
	XYZ resolution	12.5, 12.5, 2.5 micron
	Print head travel speed	30 - 300 mm/s
	Build speed	Up to 24 mm^3/s
	Build plate	Heated glass build plate
	Build plate temperature	20 - 100 °C
	Build plate leveling	Active leveling
	Build plate heat time	< 4 min (from 20 to 60 °C)
	Supported materials	Optimized for: PLA, Tough PLA, ABS, Nylon, CPE, CPE+, PC, PP, TPU 95A, PVA, Breakaway (Also supports third-party materials)
	Nozzle diameter	0.25 mm, 0.4 mm, 0.8 mm
	Nozzle temperature	180 - 280 °C
	Nozzle heat up time	< 2 min
	Build plate heat up time	< 4 min (from 20 to 60 °C)
	Operating sound	50 dBA
	Connectivity	Wi-Fi, LAN, USB port
	Language support	English, Dutch, French, German, Italian, Portuguese, Russian, Spanish, Simplified Chinese, Turkish, Polish
	Monitoring	Live camera (view from desktop or Ultimaker app)
Physical dimensions	Dimensions	342 x 380 x 389 mm 342 x 505 x 588 mm (with Bowden tube and spool holder)
	Net weight	10.6 kg
	Shipping weight	15.5 kg
	Shipping box dimensions	400 x 395 x 590 mm
Power	Required input	100 - 240 VAC / 50 - 60 Hz
	Maximum output	221 W
Ambient conditions	Operating ambient temperature	15 - 32 °C, 10 - 90% RH non-condensing
	Non-operating temperature	0 - 32 °C
Software	Supplied software	Ultimaker Cura, our free print preparation software Cura Connect, our free printer management solution
	Supported OS	MacOS, Windows, and Linux
	Plugin integration	SolidWorks, Siemens NX, Autodesk Inventor
	File types	Ultimaker Cura: STL, OBJ, X3D, 3MF, BMP, GIF, JPG, PNG Printable formats: G, GCODE, GCODE.gz, UFP
Warranty and service	Warranty period	12 months
	Technical support	Lifetime support from Ultimaker's global network of certified service partners

NOTES