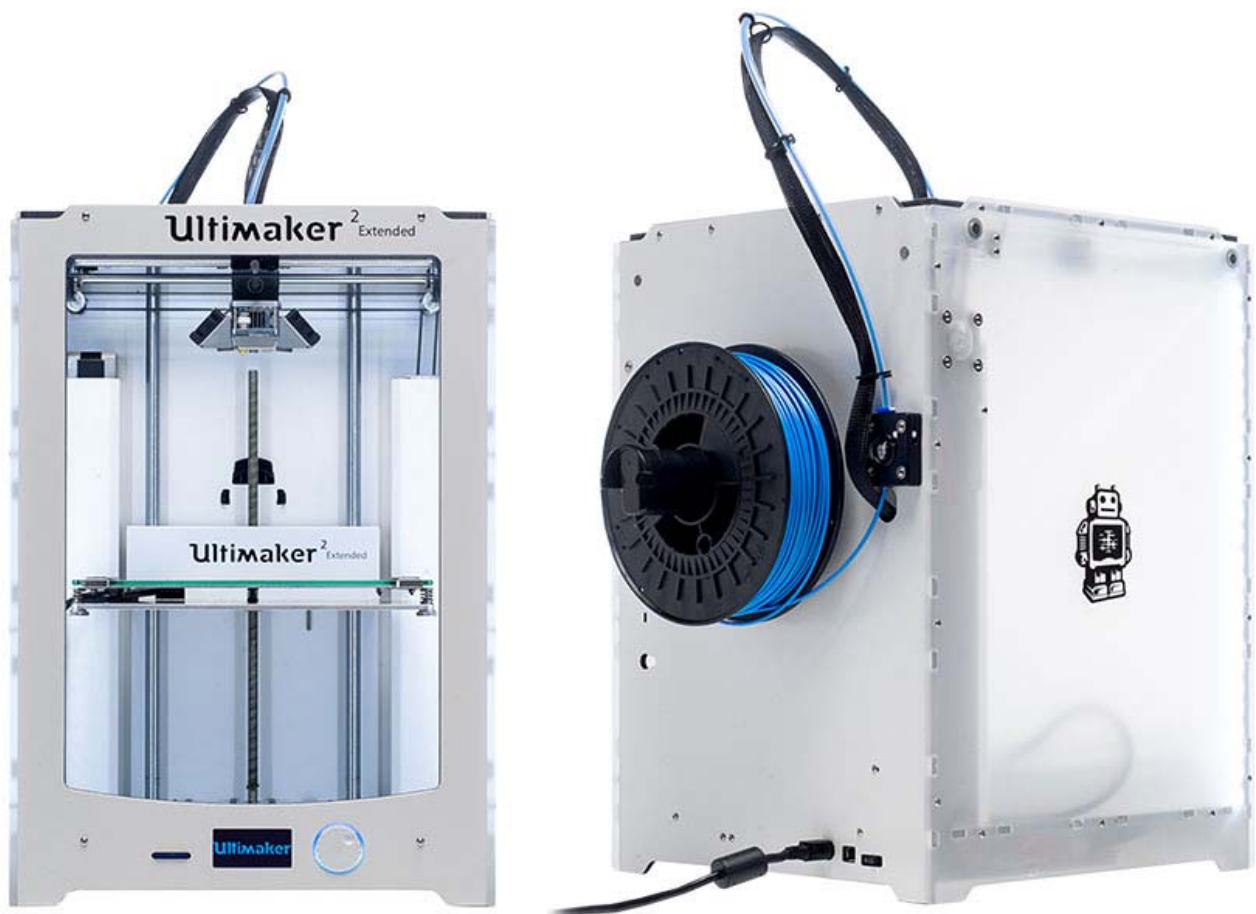




User manual for Ultimaker 2 Extended 3D printer



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

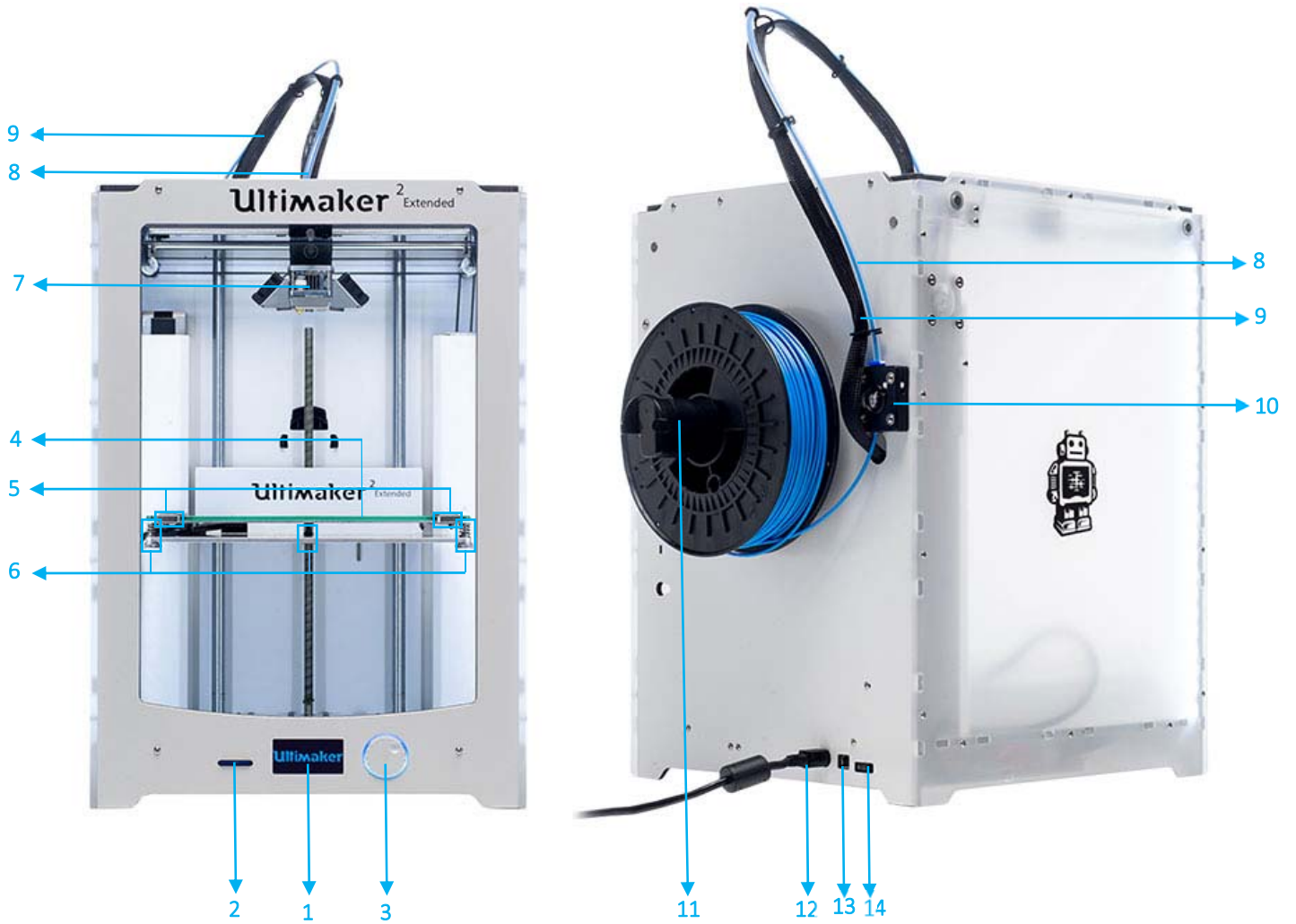
Table of Contents

- 1.0 Machine Familiarisation2
 - 1.1 Basic machine exterior set up..... 2
 - 1.2 How do they work?..... 3
- 2.0 Materials.....4
 - 2.1 PLA..... 4
 - 2.2 ABS..... 4
 - 2.3 Other material available 4
- 3.0 Where can I get the design?5
 - 3.1 Obtaining an STL File..... 5
 - 3.2 Designing a file..... 6
 - 3.3 Checking the STL file for errors 7
 - 3.4 Processing the STL file using Cura 2.1.2..... 9
- 4.0 Setting up the Ultimaker 2 Extended + 3D printer17
 - 4.1 Setting up the printer and Starting to print..... 17
 - 4.2 Finished printing 19
 - 4.3 Other Function..... 19
- 5.0 Charge forms21
 - 5.1 How does it work? 21
- 6.0 Health and Safety22
 - 6.1 What could go wrong 22
- Appendix A -Trouble shooting24
- NOTES.....25

Ultimaker 2 Extended 3D printer User Manual

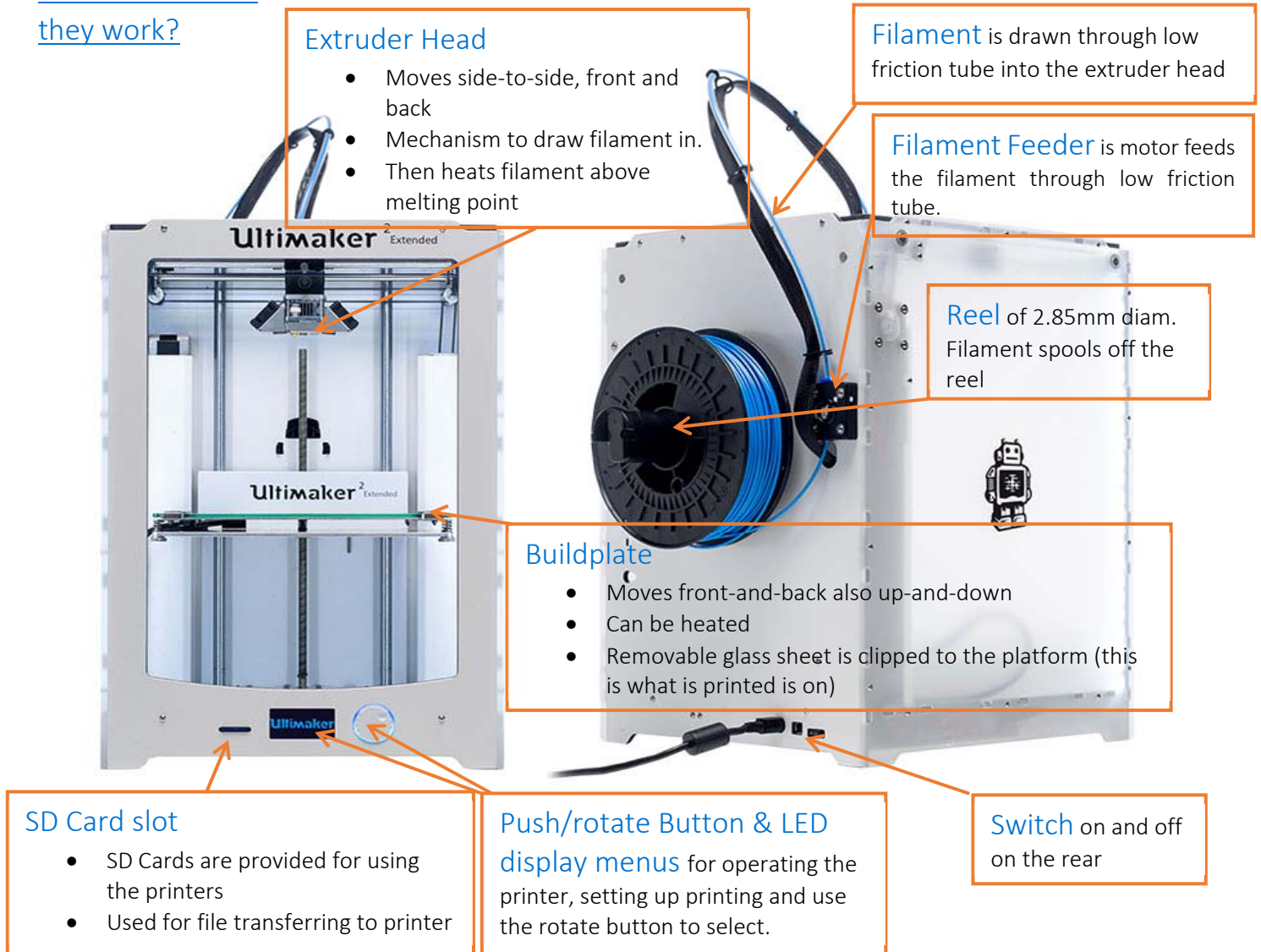
1.0 Machine Familiarisation

1.1 Basic machine exterior set up



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

1.2 How do they work?



The extruder 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 extruder 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 is 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 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
- HIPS (High Impact Polystyrene)- dissolvable so might be useful for making a mould which can later be dissolved away.
- PC (polycarbonate)
- PA (polyamide)
- PVA (poly vinyl alcohol)
- Nylon

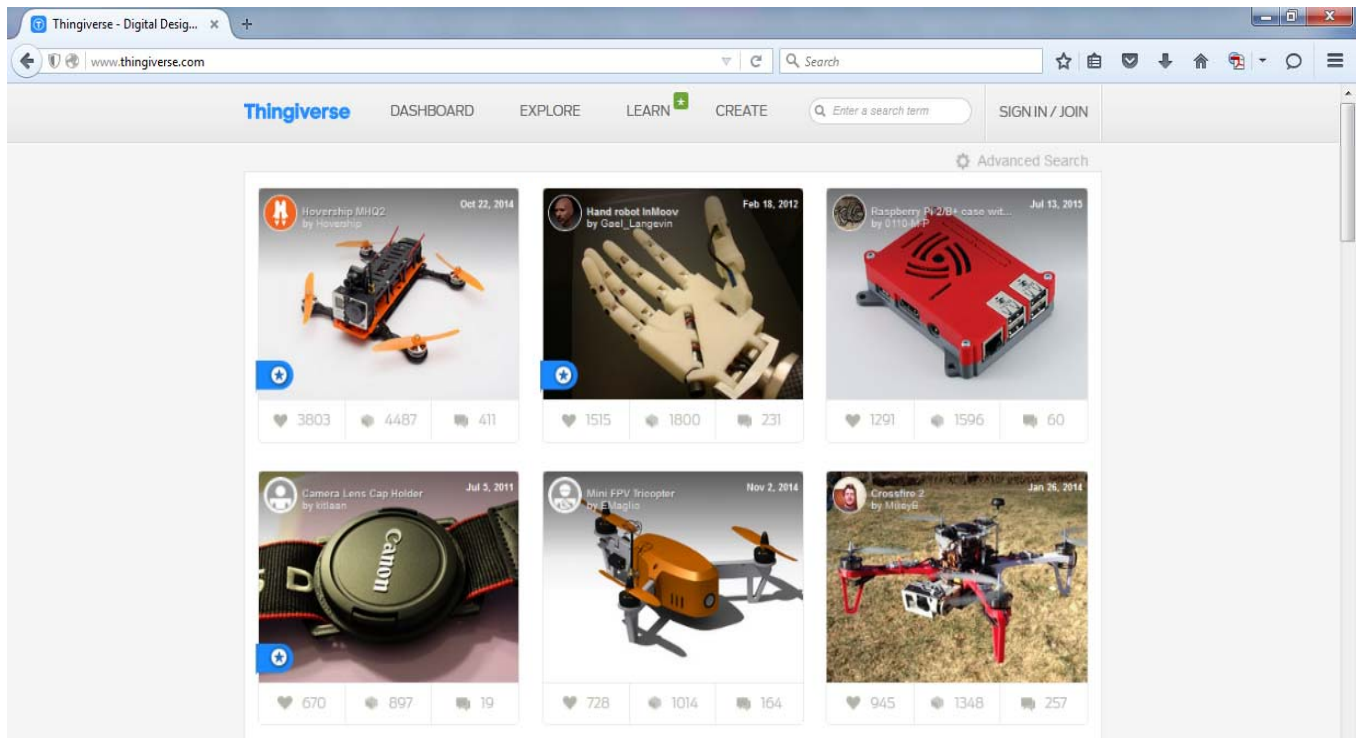
We have yet to investigate any of the above, in terms of availability (in 2.85mm diameter filament) / cost, Safety of fumes and how well our printers work with them. The printer may be harmed by using them (e.g. acidic compounds given off on heating) so further investigation is needed before using these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

[3.0 Where can I get the design?](#)

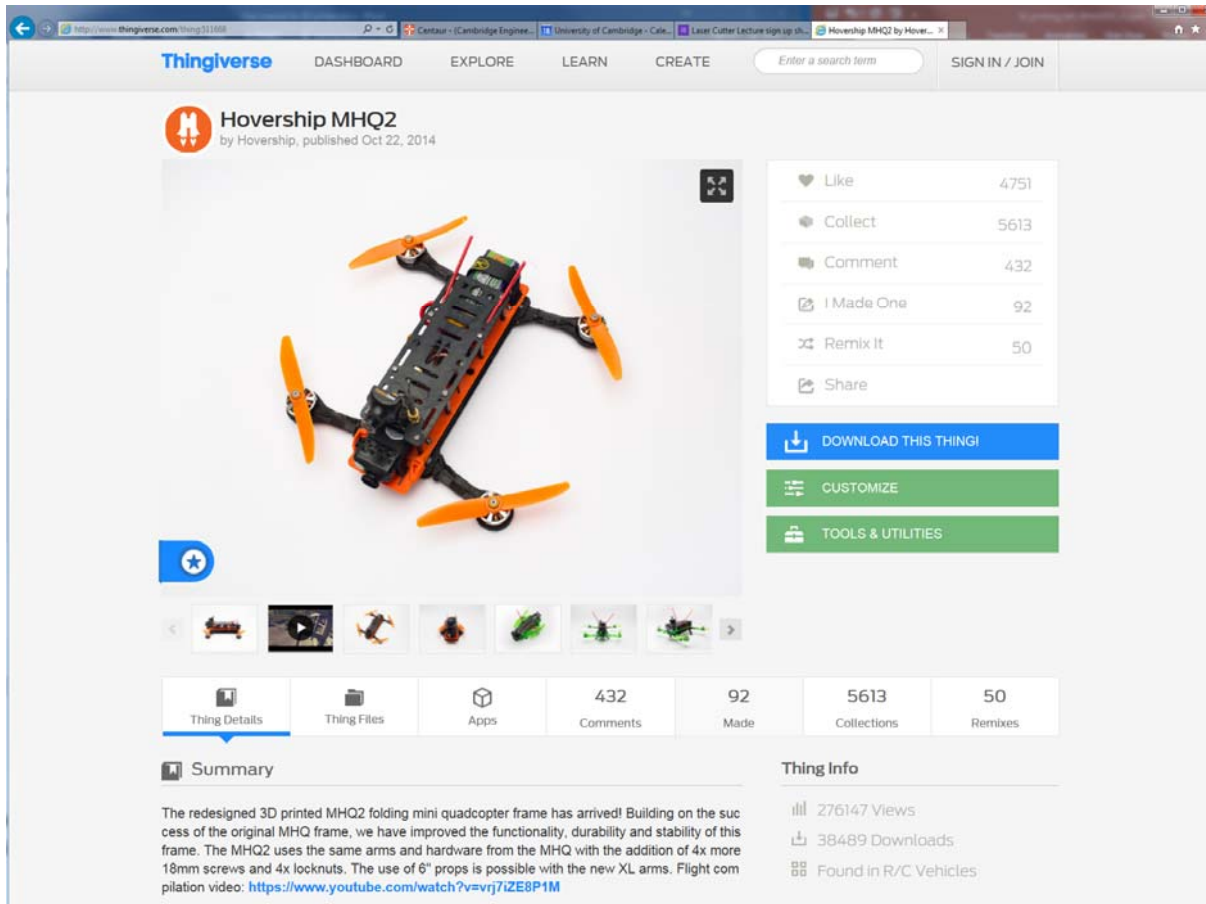
There are many online databases that offer free downloadable designs that just need to be converted into the right file format.

[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.

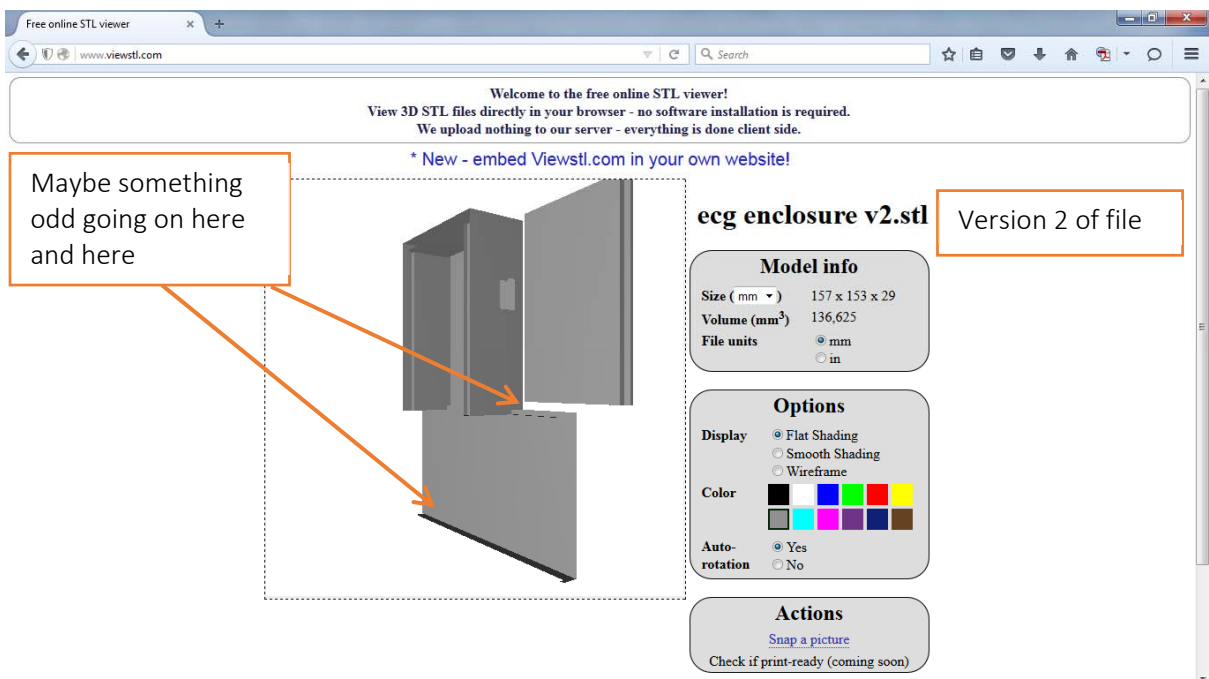
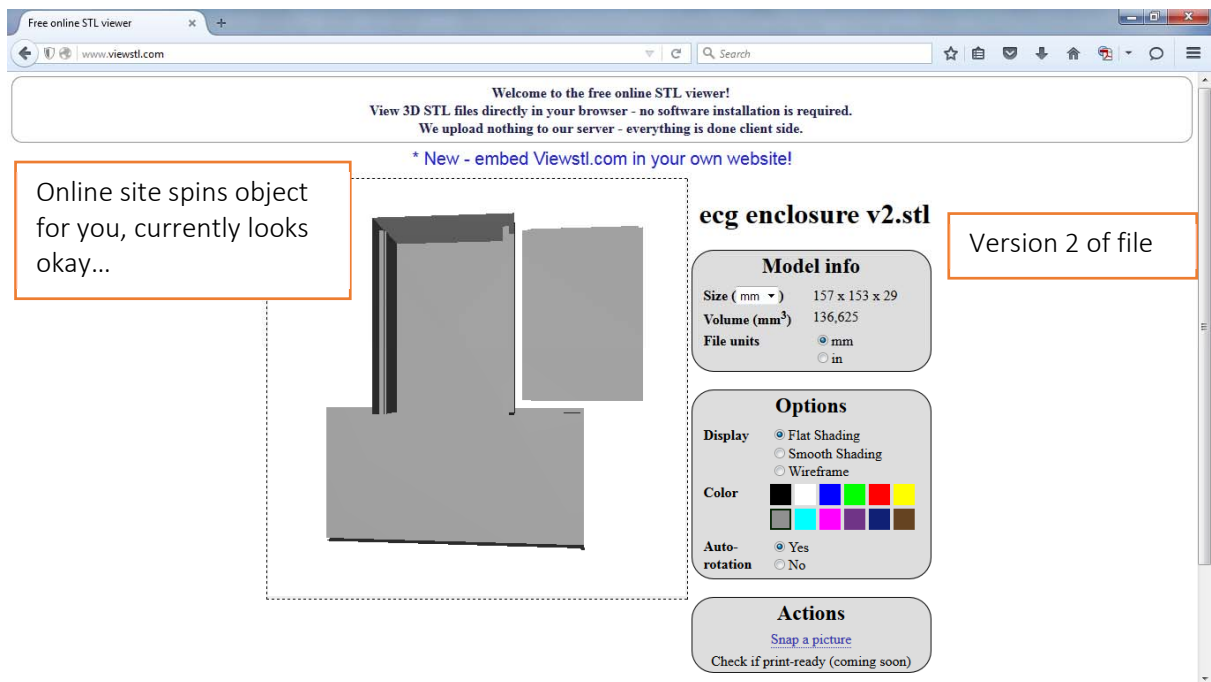
- [formerly Google's] Sketchup
- Creo (Pro-Engineer)
- Solidworks
- AutoDesk Inventor
- Catia
- IronCAD
- McNeel Rhino
- Solid Edge
- UGS NX

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

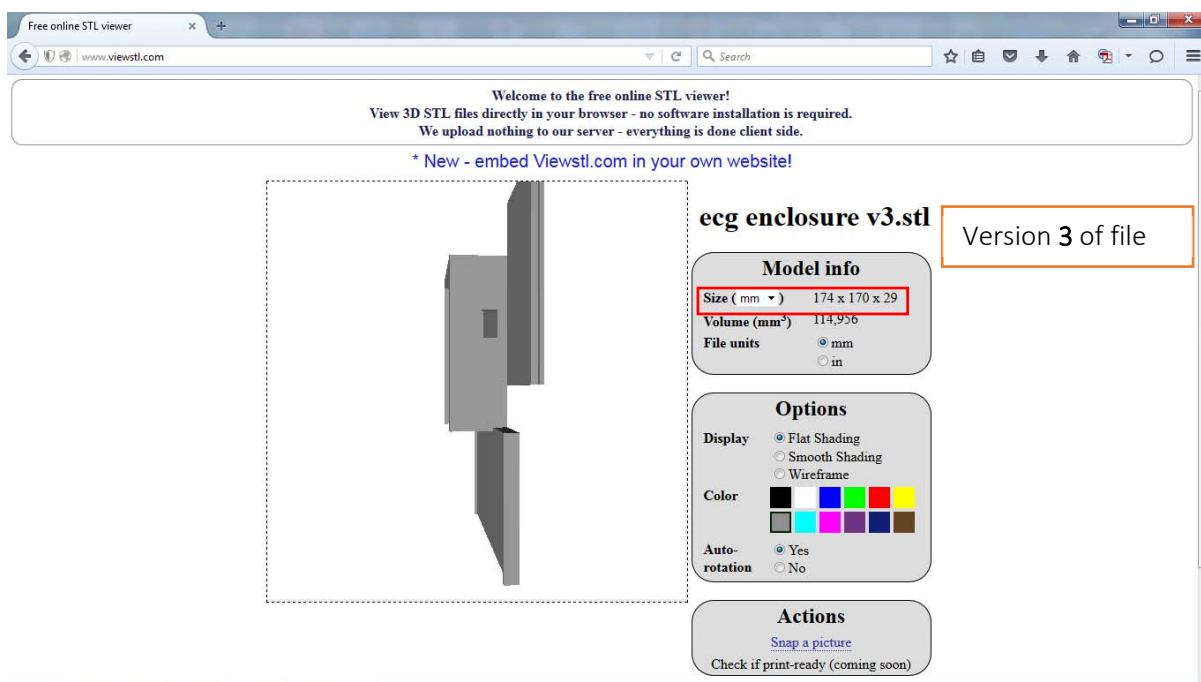
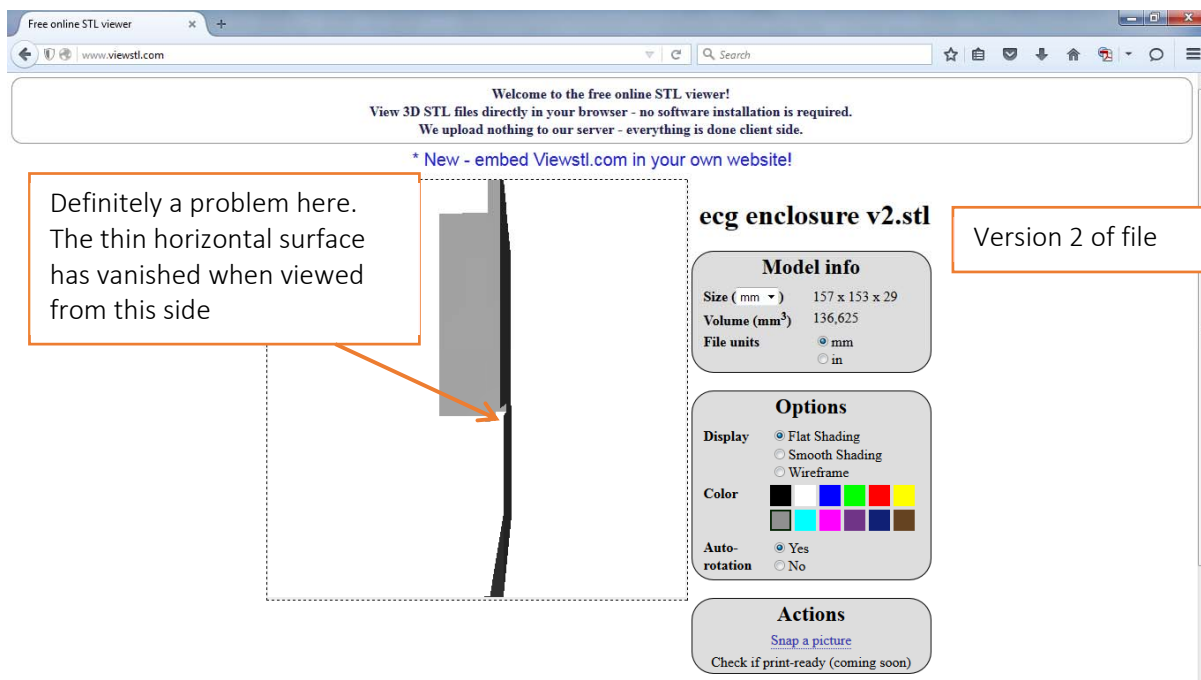
Inspiration of what to print and the uses for 3D printing designs can also be found on the Ultimaker website along with advise on using the Ultimaker software, Cura and technics in how to get a successful print.

3.3 Checking the STL file for errors

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.

WARNING: Also important to keep an eye on the size of your model, the IdeaWerk machine can print at most 150mm x 150mm x 140mm high. Note that you can print long thin parts diagonally.

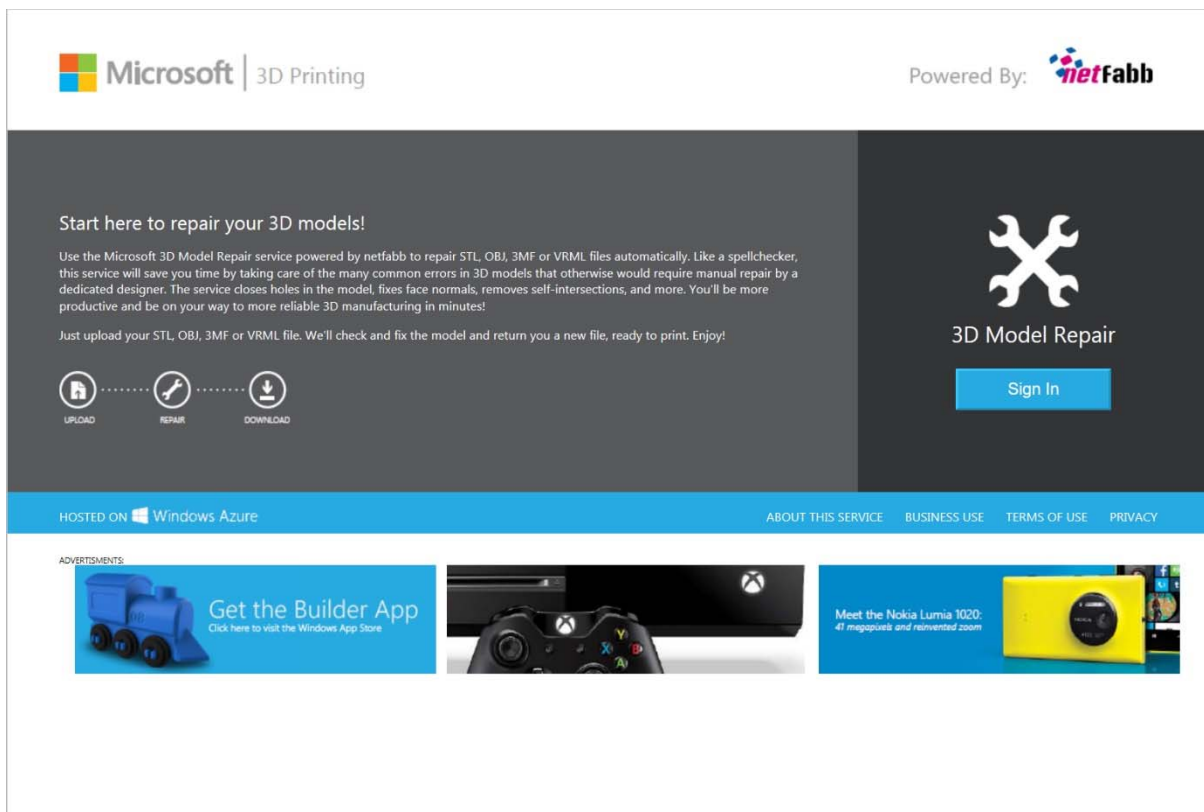
In this case, we could scale down the size of the part (see later on using DoraWare-P).

Also note that excessive scaling down can reduce part thicknesses beyond what is practically printable/durable.

3.3.1 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/>



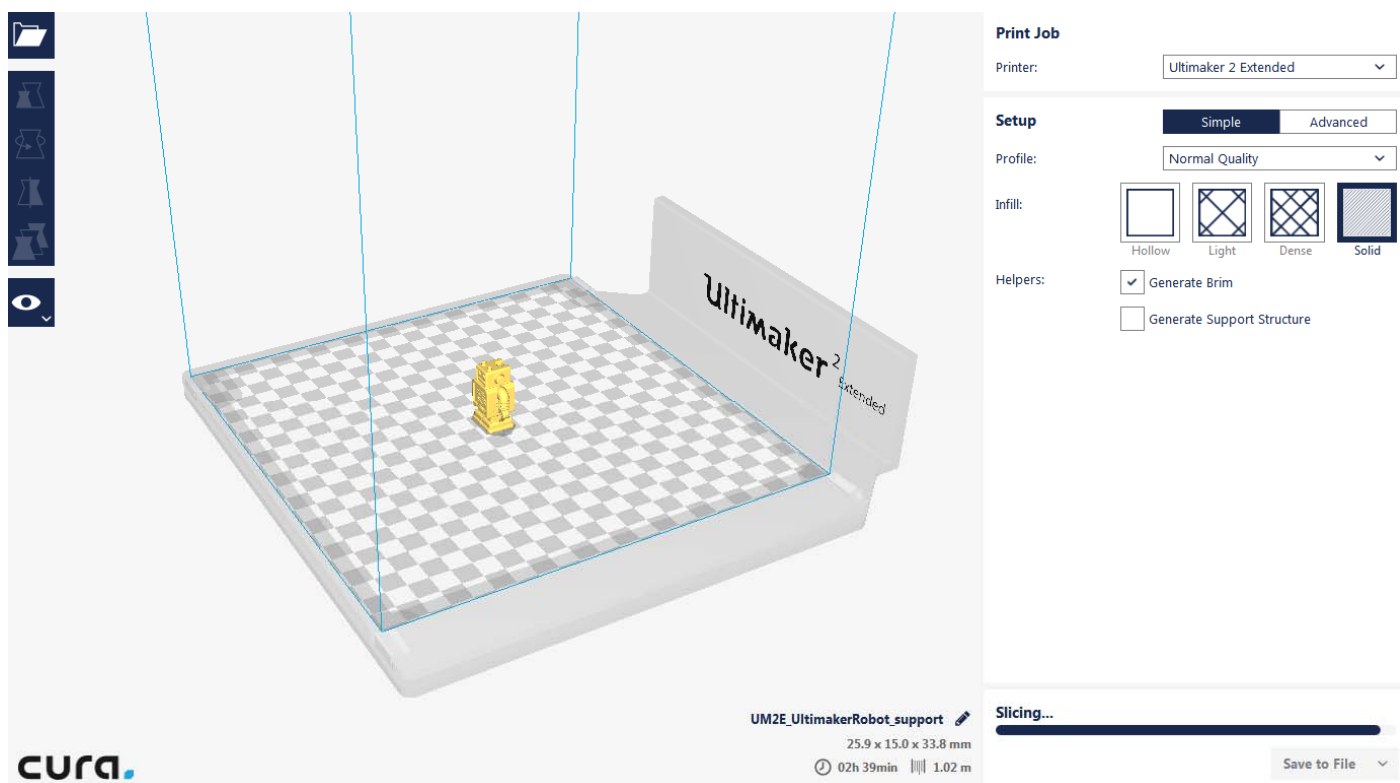
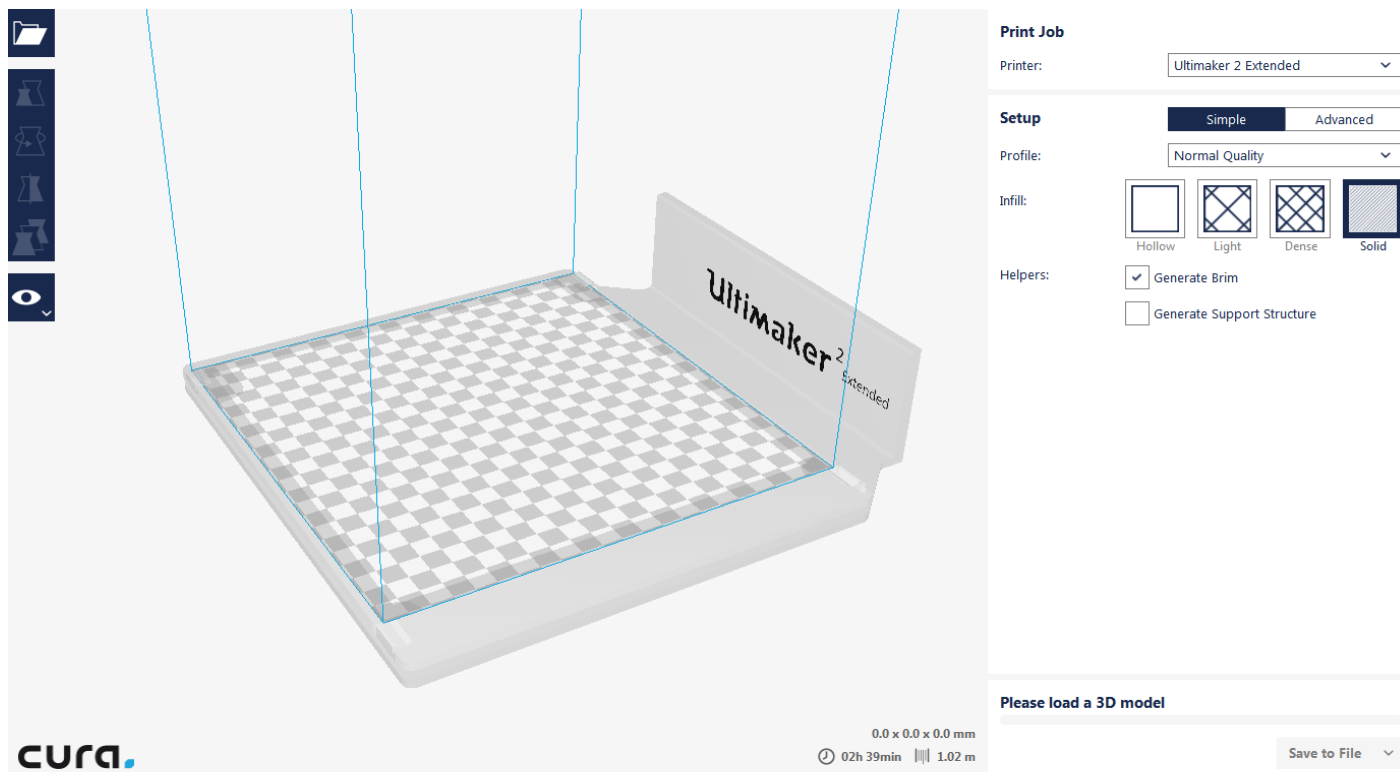
Sign in using I windows account (if you don't have an account you can create one here too) upload your STL file and click 'Repair' the site will go through your file and fix the errors, then you can down load the file. It is recommended that the file be viewed again in the STL checker. If errors still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

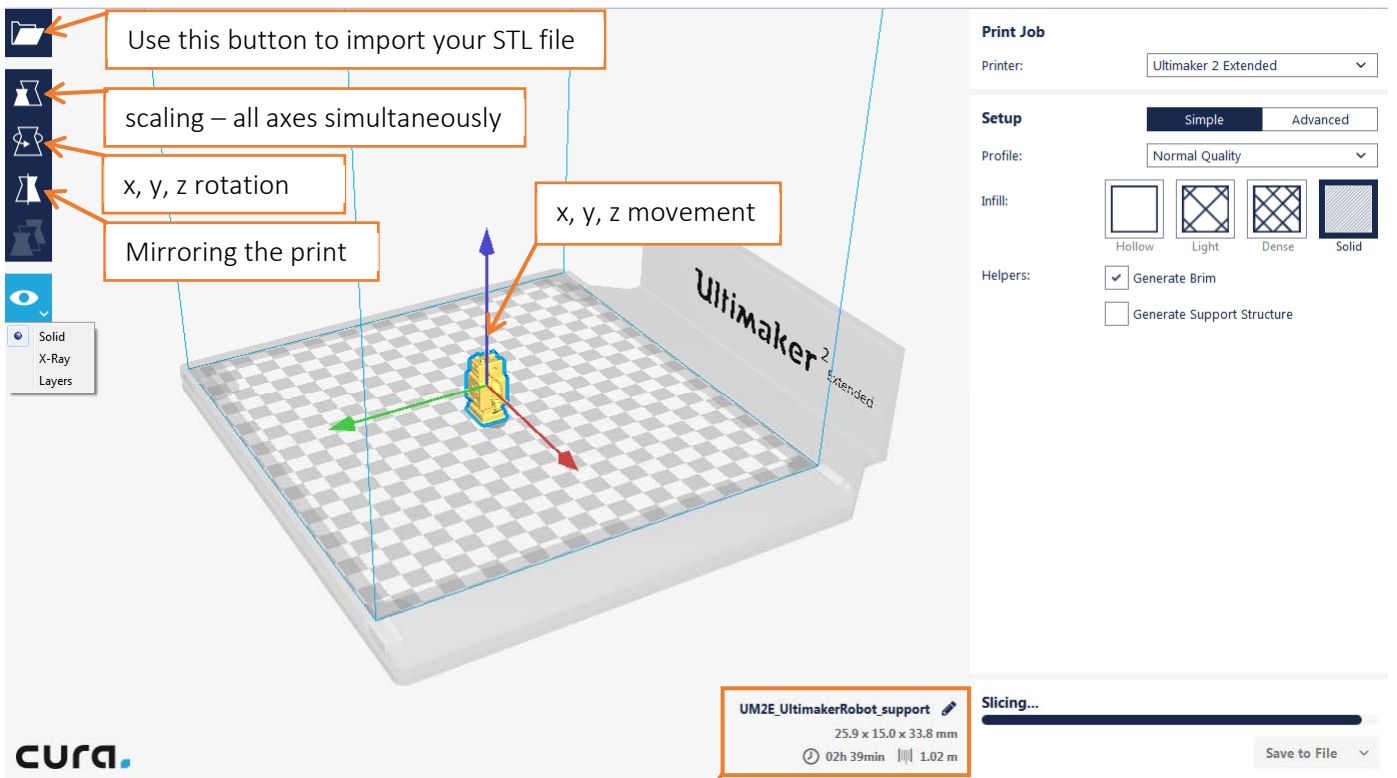
3.4 Processing the STL file using Cura 2.1.2



To process the STL file to create the G-code file which can be fed into the Ultimaker printer, we use a package called Cura 2.1.2. 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 G-code, 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 G-code can then be used to print on the Ultimaker.





Showing file name that the Gcode will be saved as, this can be edited via the pencil icon. Below is the height, width and length of the print in millimetres and last is the estimated print time and cost.

Once the print position, scaling and orientation then select the printer, the Ultimaker 2 Extended.

For simple print set up select the simple tab

Set the Profile (the quality/ detail) of the print, the higher the quality the longer and more expensive the print.

Set the Infill (the honey comb structure in the middle of the print) the higher the density the longer and more expensive the print.

Set the Profile (the quality/ detail) of the print, the higher the quality the longer and more expensive the print.

One the design is set up, scaled etc the print is then set up. The progress bar will slice the design before saying 'ready to print' now the print can be saved to the SD card to print.

Advanced options

For more options select the Advanced setup tab.

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.

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.

Setup	Simple	Advanced
Profile:	Normal Quality	
Quality	▼	
Layer Height	0.1	mm
Shell	▼	
Wall Thickness	0.8	mm
Wall Line Count	2	
Top/Bottom Thickness	0.8	mm
Top Layers	8	
Bottom Layers	8	
Infill	▼	
Infill Density	20	%
Material	▼	
Enable Retraction	<input checked="" type="checkbox"/>	
Speed	⚙️ ▼	
Print Speed	60	mm/s
Infill Speed	60	mm/s
Top/Bottom Speed	30.0	mm/s
Travel Speed	120	mm/s
Cooling	▼	
Enable Cooling Fans	<input checked="" type="checkbox"/>	
Regular Fan Speed at Layer	1	
Support	▼	
Enable Support	<input type="checkbox"/>	
Platform Adhesion	⚙️ ▼	
Type	Brim	▼
Brim Width	8	mm
Special Modes	⚙️ ▼	
Print Sequence	All at Once	▼
Experimental	▼	
Enable Draft Shield	<input type="checkbox"/>	

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. Last Skirt produces a single line around the print but not connected to it.

Special Modes

To select the print sequence and whether to print all objects one layer at a time or to wait for one object to finish before moving onto the next. One at a time mode is only possible is all models are separated in such a way that the whole print head can move in between and all models are lower than the distance between the nozzle and the X/Y axes.

Experimental

Adjusting Enable draft shield will create a wall around the object which traps (hot) air and shields against exterior airflow. Especially useful for materials which warp easily. Adjusting the Conical Support will make support areas smaller at the bottom than at the overhang.

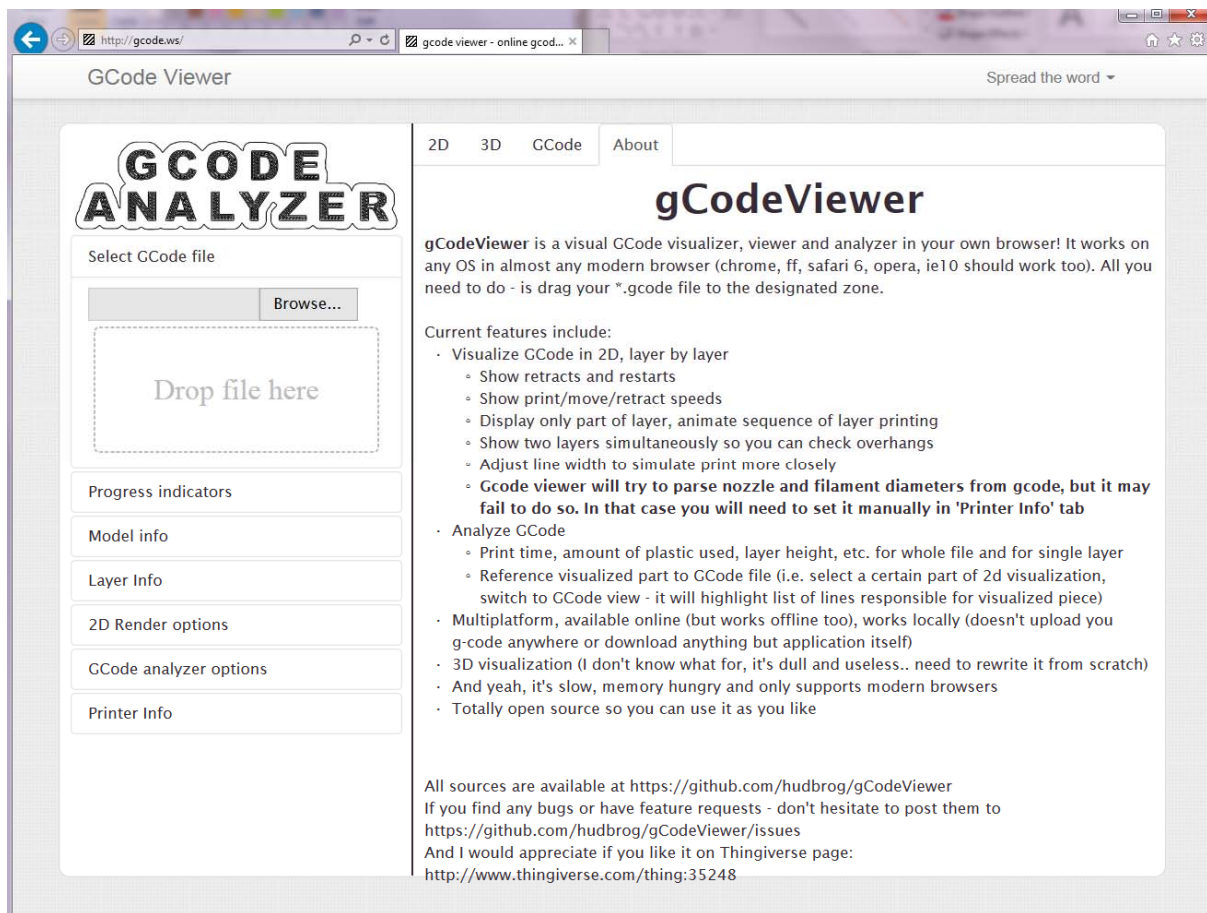
NOTE: Temperature setting are set automatically

3.4.1 Saving the file to Gcode file format to print

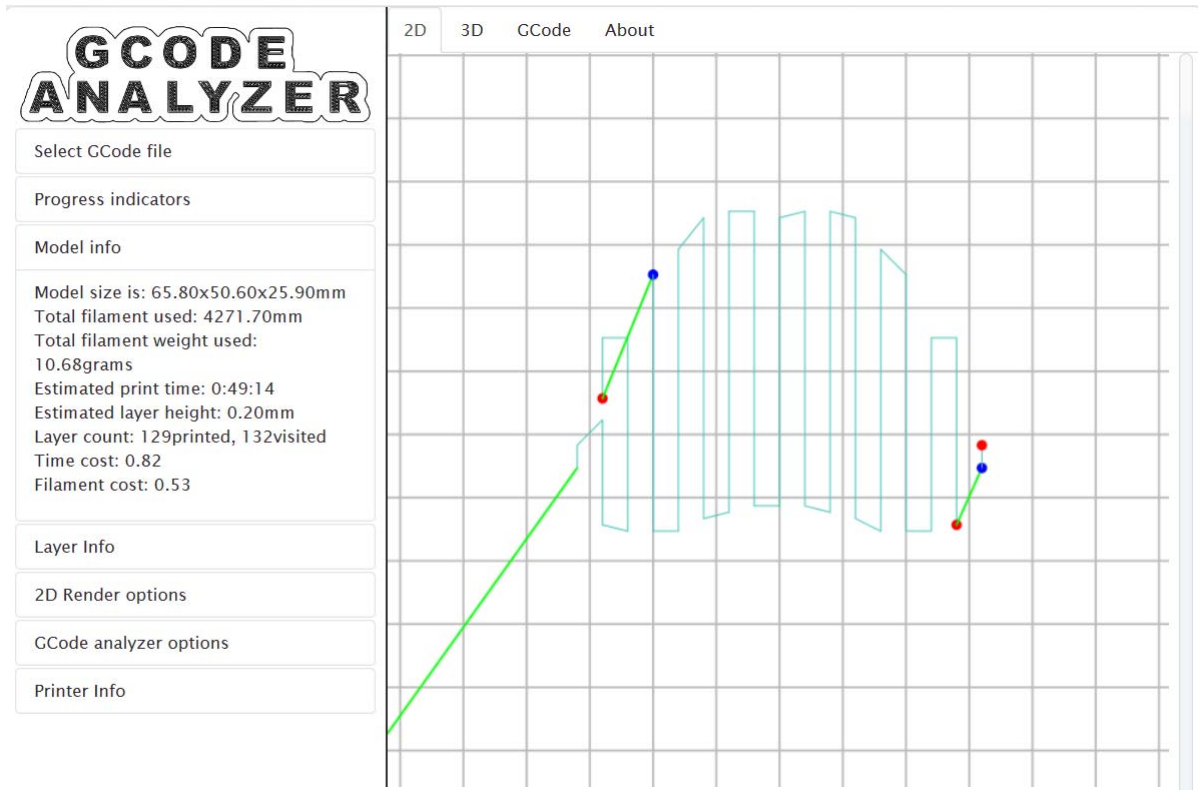
Click 'save to file' to generate the G-code, the progress bar will then start to show bar as it processes each lay of the design. Note that this is computationally fairly complex and that time taken may vary from a few minutes for something simple to about half an hour for the example item with no support 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.

Tracking the problem with the G-code

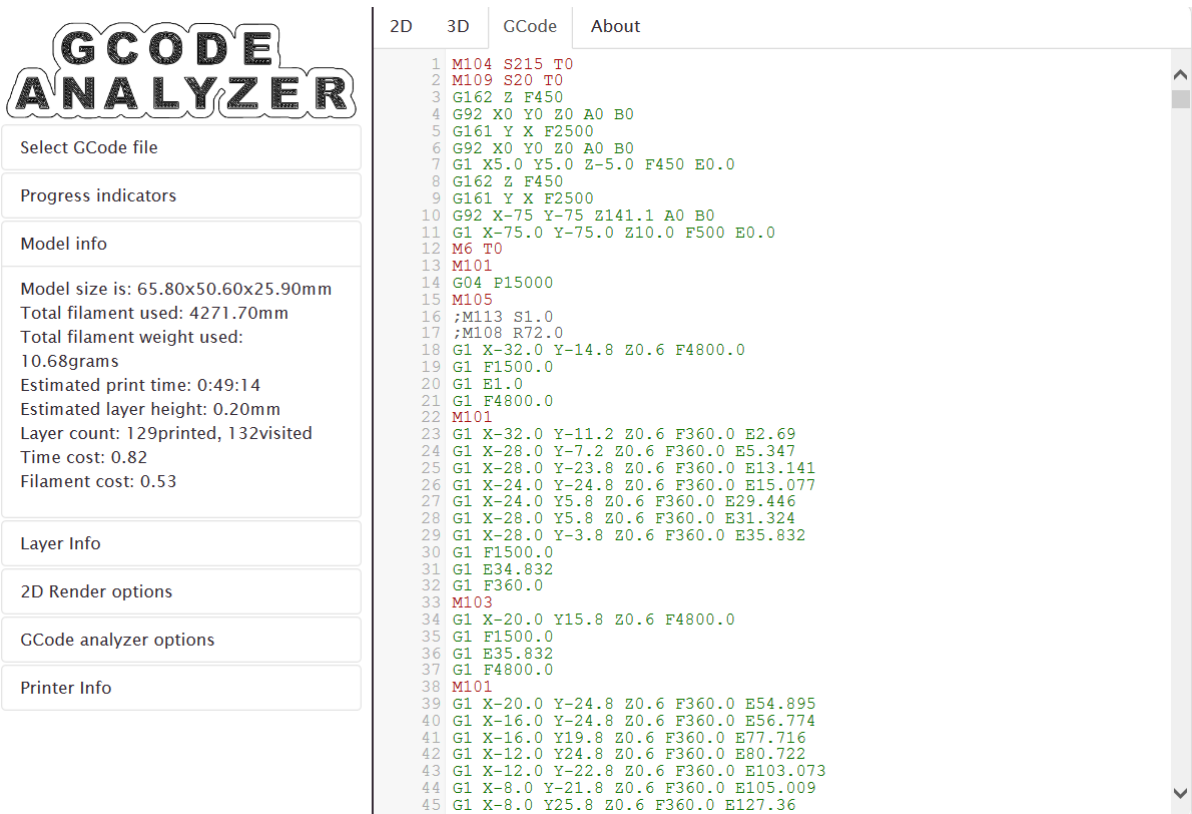
Sometime when producing the G-code, the print does not turn out the way you expect i.e. holes in gears are filled in. You can check that the problem is not with the G-code via GCODE ANALYZER, go to <http://gcode.ws/>



a. Upload your G-Code



b. Looking at the 2D tab will show the root the raft will be 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**.

GCODE ANALYZER

Select GCode file

Progress indicators

100%

55%

Model info

Model size is: 65.80x50.60x25.90mm
 Total filament used: 4271.70mm
 Total filament weight used: 10.68grams
 Estimated print time: 0:49:14
 Estimated layer height: 0.20mm
 Layer count: 129printed, 132visited
 Time cost: 0.82
 Filament cost: 0.53

Layer Info

Layer number: 0
 Layer height (mm): 0.6
 GCODE commands in layer: 54
 Filament used by layer (mm): 324.28
 Print time for layer: 122.9sec
 Extrude speeds:
 ■ = 25.00mm/s
 ■ = 6.00mm/s
 Move speeds:
 ■ = 80.00mm/s
 ■ = 25.00mm/s
 ■ = 6.00mm/s
 Retract speeds:
 ● = 25.00mm/s

Printer Info

Plastic diameter:

Plastic type:
 ABS
 PLA

Nozzle size:

Printer hourly cost:

Filament price(per gram):

GCode analyzer options

These require re-analyzing file:

- Sort layers by Z
- Hide empty layers
- Show GCode in GCode tab (memory intensive!)

2D Render options

Speed display type:

- mm/sec
- mm extrusion per mm move
- mm³/sec
- Show non-extrusion moves
- Show retracts and restarts
- Move model to the center of the grid
- Show different speeds with different colors
- Emulate extrusion width
- Render lines slightly transparent
- Show +1 layer

GCODE ANALYZER

Select GCode file

Progress indicators

Model info

Model size is: 65.80x50.60x25.90mm
 Total filament used: 4271.70mm
 Total filament weight used: 10.68grams
 Estimated print time: 0:49:14
 Estimated layer height: 0.20mm
 Layer count: 129printed, 132visited
 Time cost: 0.82
 Filament cost: 0.53

Layer Info

2D Render options

GCode analyzer options

Printer Info

2D 3D GCode About

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

4.0 Setting up the Ultimaker 2 Extended + 3D printer

Now that the Gcode file has been created the printer can be set up, before using please check:

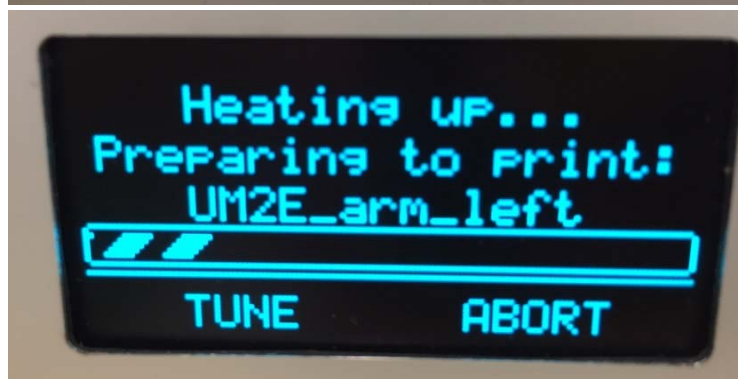
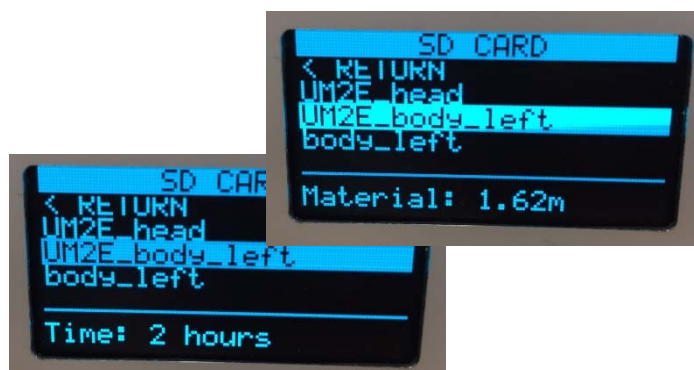
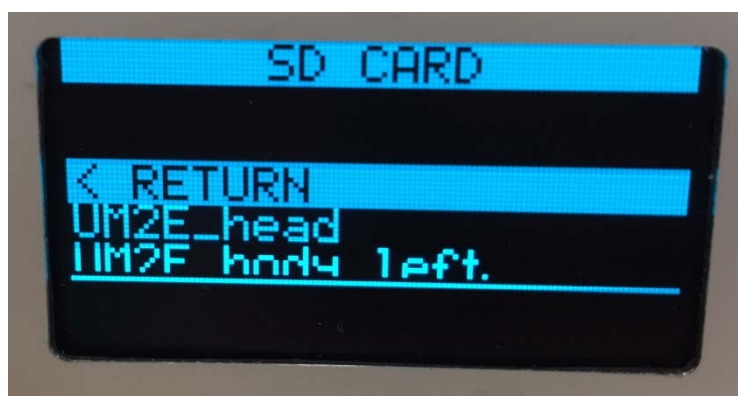
- The printer is on (at the wall and at back).
- Also that a glass bed is clipped with four clips to the aluminium platform.
- Check that the printer is loaded with the correct filament (both type of plastic and colour) located at the back of the printer and that there is enough filament to complete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

4.1 Setting up the printer and Starting to print

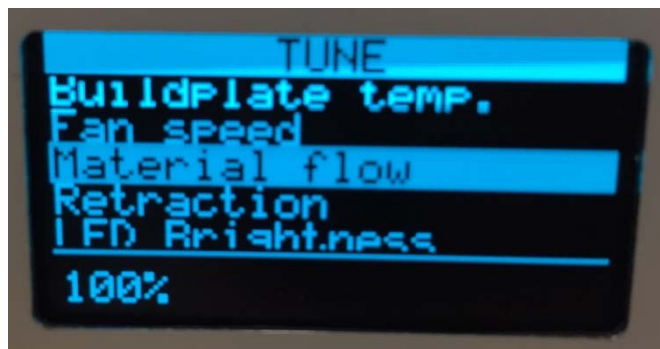
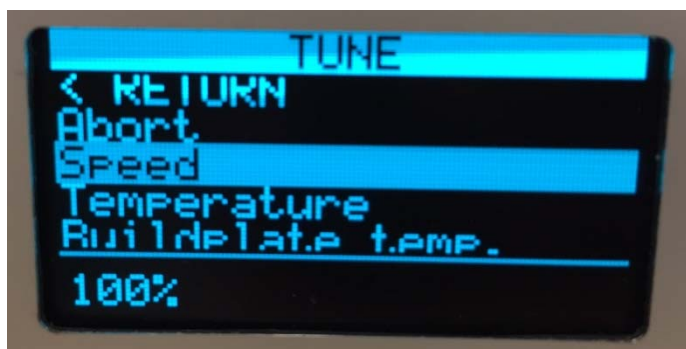
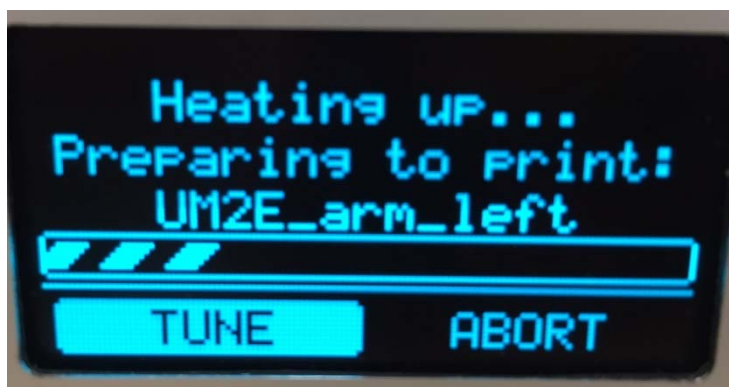
When the printer is first powered up the start-up screen will appear as shown below



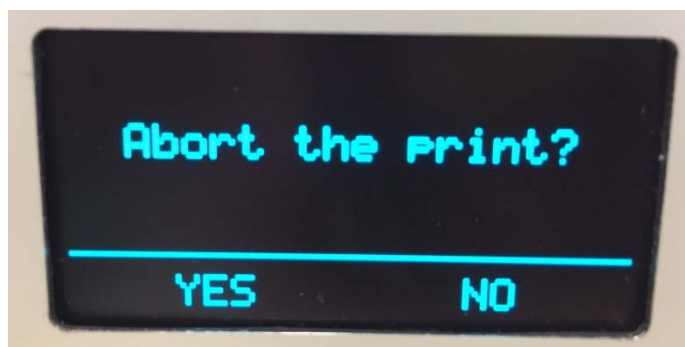
Insert your SD Card in to the SD card slot located to the left of the screen, now use the glowing dial to the right of the screen to select the 'Print' option (push the dial to select). Select the file you want to Print and push the dial to confirm, along the bottom information such as the print time and the material size scroll along for each highlighted print.



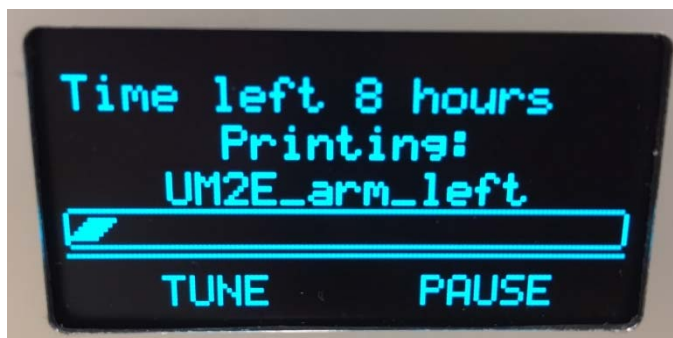
Once the print has been confirmed the printer will start to heat up, at this point the printer head will move into position ready to print. By selecting 'Tune' the printer settings can be modified such as print speed and build plate temperature.



There is also an option to abort the print, if selecting this the printer will then ask 'Abort the print?'



However, if you just wish to pause the print, once the print has started there is also an option for this too.





[4.2 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 you can the display shows the time of the prints progress and how long is left.

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) pull/rotate the front two clips off the glass plate and slide the buildplate out from the printer. When the printed item is cooled to room temperature, it should be possible to pull it off the bed, may need to use the spatula to remove the print, take care not to mark or crack the glass. Please then replace the plate and clips and don't forget to pay what is owed for the cost of materials.

[4.3 Other Function](#)

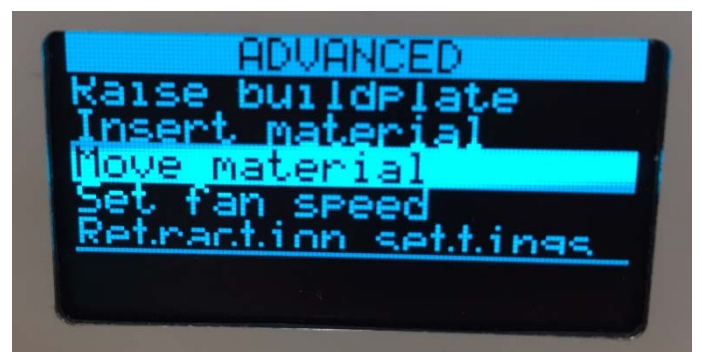
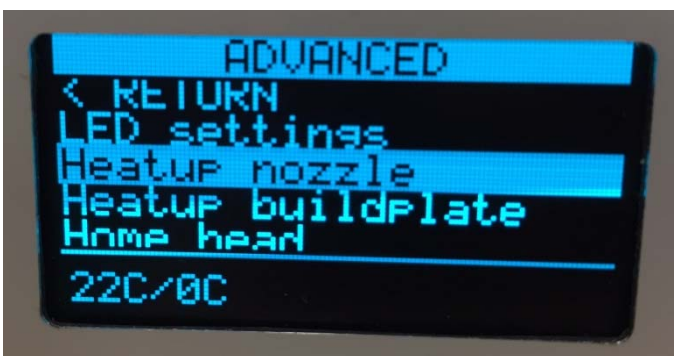
The other option on the start up screen should only be accessed by fully trained personnel.



The 'Materials' option will allow you to 'Change' the material type and therefore the default buildplate and extruder temperatures.



'Maintenance' also allows you to move and heat the head and the buildplate along with other functions.



5.0 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.

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.

5.1 How does it work?

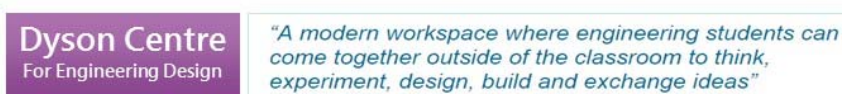
Please use the electronic scales to work out the mass of PLA used, include any scrap, rafts, supports and all failed attempts (these are at your cost). **The charge is £1 per 20g of part there-of**, for example.

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

Upon collection please take this to the Dyson centre desk to pay



Dyson Centre 3D Printing Charging Sheet

Machine	Mass used, including wastage (grams)	Cost per gram (£/gram)	Total cost (£) ie whole pounds only
RS Ideawerk Pro/Plus (WT280A)	Use scales provided to weigh parts	£1 per 30grams , ie 0-30grams: £1 30-60grams: £2	
Ultimaker 2 Extended+	Use scales provided to weigh parts	£1 per 20grams , ie 0-20grams: £1 20-40grams: £2	

<input type="checkbox"/>	Cash	Amount:£ ____
<input type="checkbox"/>	EPOS (payment will be recorded and check)	Amount:£ ____
<input type="checkbox"/>	Job number (please enter job number):	Amount:£ ____

Contact Information

Name: _____ Date _____

Email: _____ Tel: _____

Signature: _____

Cash Receipt For £ _____

Item description: _____

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 manager (rlr20) before use:

- We may be prepared to allow limited use...
- ...as long as there isn't a queue of people waiting to use the machines...
- ...and **subject to an additional charge equivalent to 1% of machine cost**
- **RS Ideawerk an addition £7 per print up to £7 i.e. print is 20g therefore cost is £8 (£1 for the print + £7 for usage**
- **Ultimaker 2 Extended + an additional £28 per print up to £28 i.e. print is 19g therefore cost is £29 (£1 for the print + £28 for usage**

Dyson Centre
For Engineering Design

6.0 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

6.1 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 SD card, redo the Gcode and resave the file on to SD card.

Problem: I find my file on the SD when I plug it in to print.

Solution: The file is not in the right format.

NOTES