
ULTIMAKER

This report imparts the experiences of a 3rd Year Industrial Design student printing with the Ultimaker 3D printer. The report also discusses the highlights and limitations of the printer and furthermore the improvements needed for the printer to become commercially available to the mass market.

Cura

Desirable Preset for a normal print quality;
Layer height: 0.2mm
Fill: 20%
Speed: 60mm/s
Temperature: depends on pigment of filament, the room temperature etc.

Limitations of Cura

Cura is limited with options and customization. When applying a brim platform adhesion, the user cannot choose which points of the model are encompassed within the brim. This can cause a weak point of adhesion to the bed and cause the model to wobble during print.

The user cannot choose the starting height of the bed, it is preset and this limits the flexibility of the machine. The nozzle is not shown in the preview. If there was a bed height option, the nozzle would need to be shown for reference.

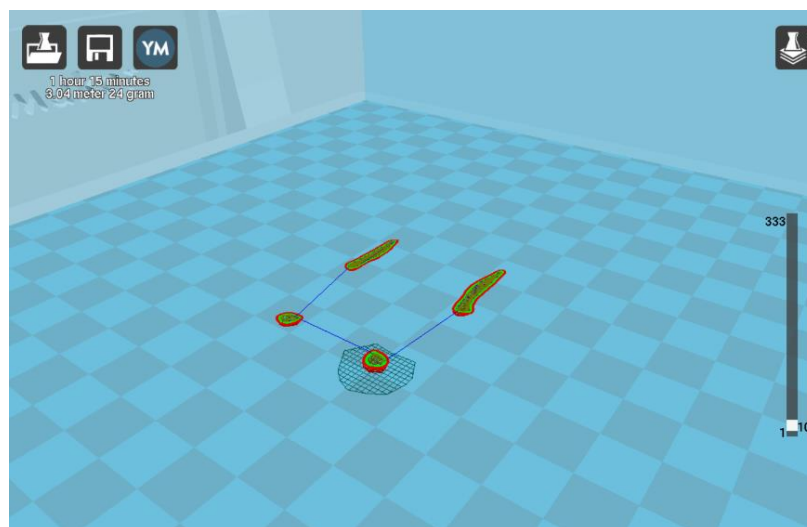
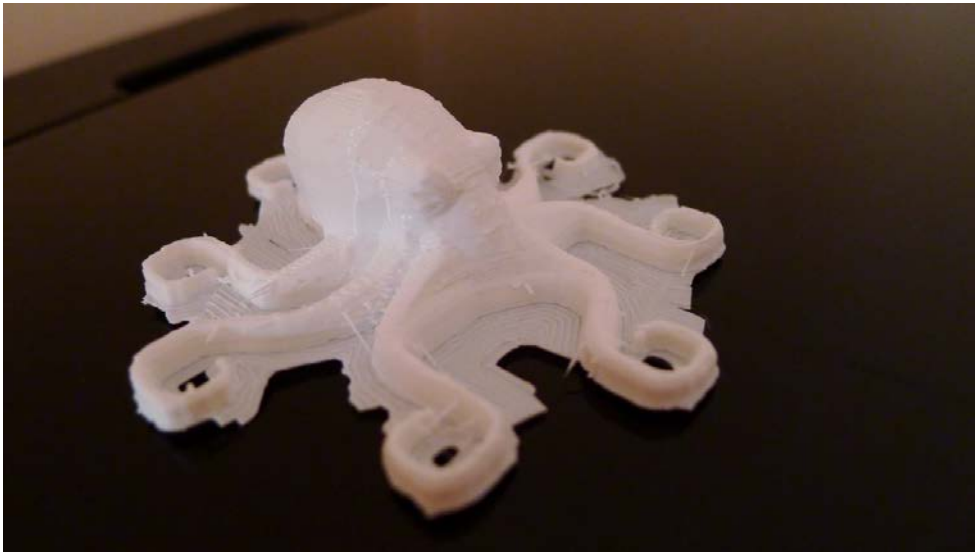


Figure 1 - As you can see, the brim has been applied to only one point of contact with the print bed.

Interesting Prints

I experimented with printing many different forms however below I focus on organic shapes and explain why some prints failed more than others.

OCTOPUS SMALL - 50% INFILL 60MM/S 0.2MM LH ADHESION:
BRIM

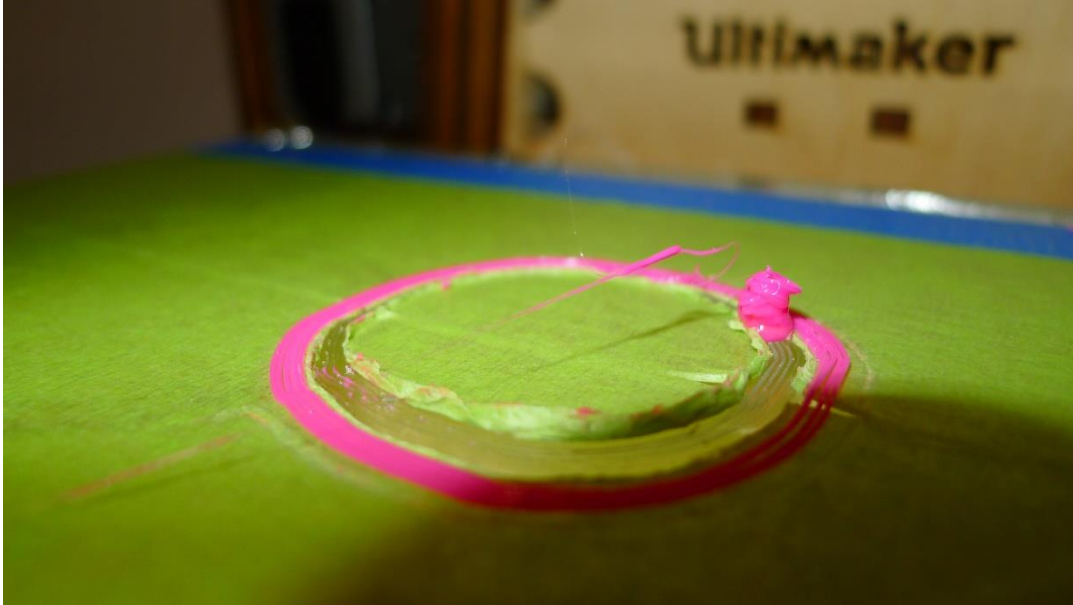


OWL - 40% INFILL 60MM/S 0.15 LAYER HEIGHT ADHESION: BRIM



TOTORO DOLL - 30% INFILL 60MM/S 0.1 LAYER HEIGHT 220DEG

This print was peculiar as it failed 7 times. The totoro failed 3 times due to the fan having too high RPM, the fan box being too low and scraping the top layers and the masking-tape type. Some masking tapes leave a residue of glue. This is could be due to the glue retaining heat more easily than other masking tapes.



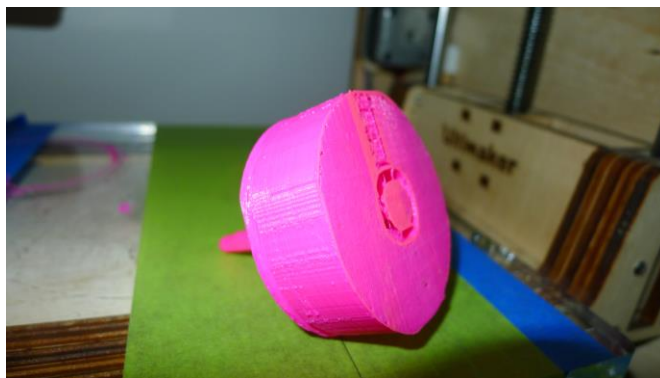
TOTORO - 10% INFILL 60MM/S 0.1 LAYER HEIGHT 215DEG ADHESION: BRIM



FLOWER - 10% INFILL 60MM/S 0.15 L/H 215DEG ADHESION: BRIM



SPINNING TOP - 10 INFILL 60MM/S 0.15MM LH 215 DEGREES,
ADHESION: RAFT AND SUPPORT TOUCHING BUILD PLATE



Unfortunately, I cannot free the spinning top from its build plate support as the support infill percentage is higher than the infill of the actual model. This is another limitation of Cura. You cannot customize the supports.

CAT – 10 INFILL 60MM/S 0.15MM LH 215 DEGREES, ADHESION:
RAFT AND SUPPORT TOUCHING BUILD PLATE



This model failed due to the four points not all being included in brim. Below you can see that the nozzle has moved a weaker point and sufficiently ruined the foundation of the model.



Limitations of Ultimaker

The user interface is limited in its options. The user cannot control the x, y, z axis whilst printing, even if paused. This can be bothersome especially if the user wishes to adjust the bed height.

If adjusting the fan box, the nozzle may move out of place, causing the printer to need total re-calibration and if this doesn't work then the user must manually readjust the screws directly above the nozzle. If the bed seems too low, it may be that the nozzle is not in an optimal position and needs manual re-adjusting.

Aesthetically, the user interface can be intimidating and confusing to use if the user is not instructed prior to use. The UI is far from intuitive and this will hinder its commercial success.

The first layer is almost always in need of supervision, as 5 times out of 10 it fails due to an array of possible reasons. Troubleshooting is a large part of the Ultimaker and unfortunately inevitable with its current level of technology, however at its price point, it is one of the best on the market at its time of release.

A big problem with the ultimaker, is the fan box. The box melted when it came close to contact with the nozzle, which wasn't too surprising considering it is made of plastic. The box fell apart several time during print, ruining the whole print. I had to wrap electrical tape around it to keep it intact and lift it higher than the nozzle, as the box scraped against each layer the printer laid down. Eventually, I printed a new fan box courtesy of Thingiverse and attached via the existing screws. It is thinner and a solid uni-body.

PLA

When changing filament color print a test cube 3x3 at 60/70mms with a temperature starting at 215 degrees, then on the fly change the temperature to adjust print quality.

A peculiar observation, the pigment of filament may affect heat retention for example, 215 degrees Celsius is optimal for pink, anything higher starts to burn the filament and leaves a sloppy surface finish on the model. Burnt pink filament looks like a slightly fluorescent version of the color you are printing in.

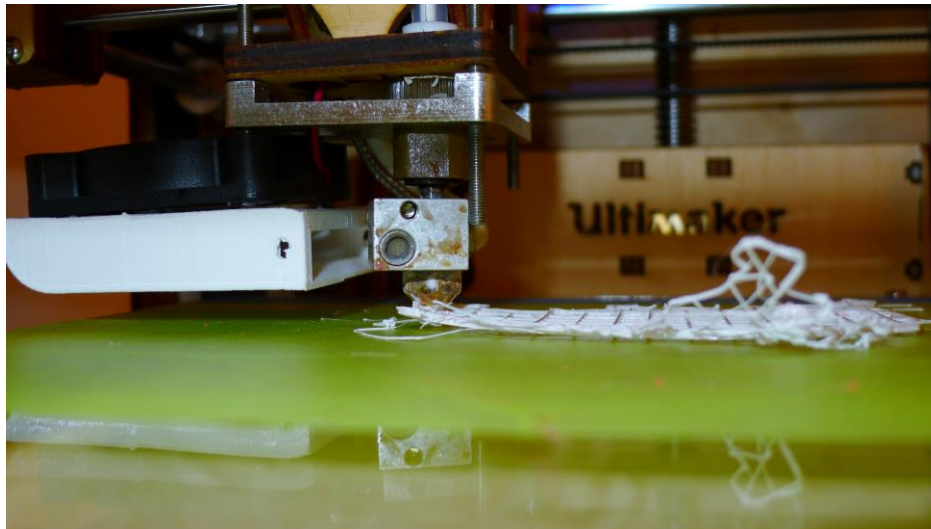


Figure 2 - The initial layer has bowed and distorted possibly due to the nozzle temperature not being high enough. This causes the filament to set before it has had a chance to adhere to the bed.

Summary

The experience has made me appreciate design and model making on a whole new level. I found that I can enjoy the concept stage much more since I can print concepts that have potential and save a lot of time. The experience has also however shown me the downfalls of 3D printing. If there is a mistake in the design, you will not find out usually until the print has finished whereas if you are making the model yourself, you can change and tweak the design before the model is final. 3D printing is efficient however currently, it is not very flexible. The experience of 3D printing has put to my attention that it is much more accommodating to people with respiratory issues whereas model making with MDF or BlueFoam is much more strenuous on health and time.

By Katherine Alicja Ewa Kawecki