How to 3D Print at the DSC

A guide for new and current staff members at the DSC. This document first goes over the basics of 3D printing, then presents some specific problems and typical prints.

1 Choosing a Printer

Ask yourself the following questions:

1. Does the object have a large footprint)?
2. Is the object thin (< 1 cm)?
3. Is the object really detailed?
4. Do all surfaces of the object need to be smooth?
5. Is it a two-colour print?
6. Has the object been requested to be made out of TPU 95a?

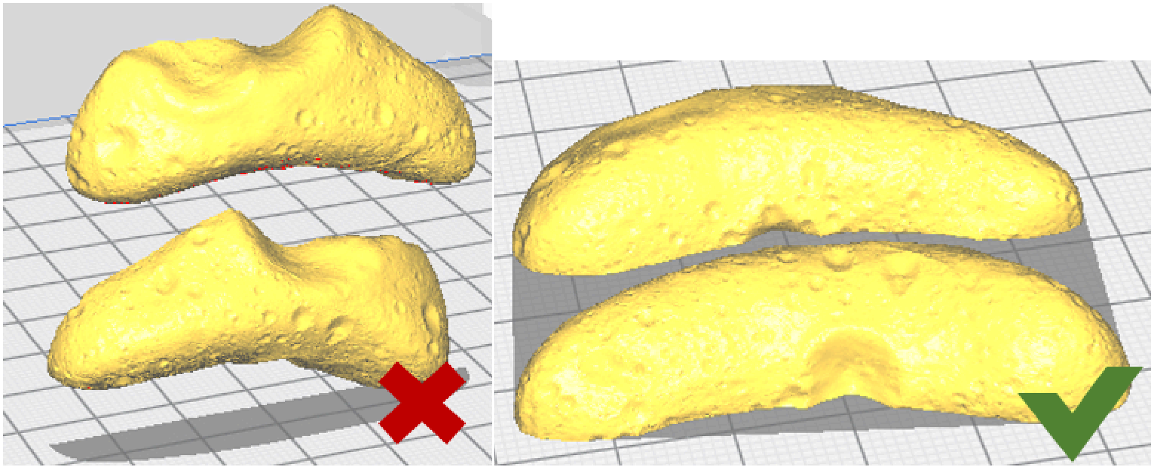
If the answer to any of these is yes, it should be printed on an Ultimaker. The Ultimakers are used for 99% of all DSC jobs, while the Makerbot is used for demonstrations and at very high volume periods only.
If the answer to question 6 is yes, it should be printed on one of the Ultimaker S5.

2 Setting Up a Print Job

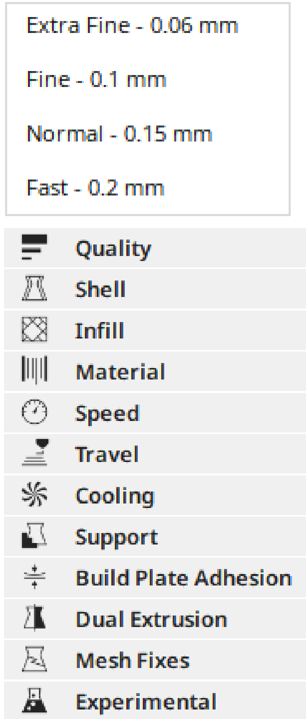
Cura is used to set up prints for the Ultimaker 3s and S5s.

Cura

1. Open Cura
2. Near the top-left corner, click on the folder icon . Open the file(s) to be printed
3. Change the position of the object on the build plate.
    a. Click on the object such that it's surrounded by a blue outline and drag it around.
    b. Once other tools such as Rotate are used, the Move tool must be selected from the left toolbar to be able to click and drag the object around again.
    c. To move the object in only one direction, click and drag on the arrows instead of the object.
4. Change the orientation of the object(s) (if necessary)
    a. Click on the object to be rotated. Selected objects will be surrounded in a blue outline.
    b. On the toolbar at the left side of the screen, click on the Rotate icon . Click and drag on the rings to rotate the object.

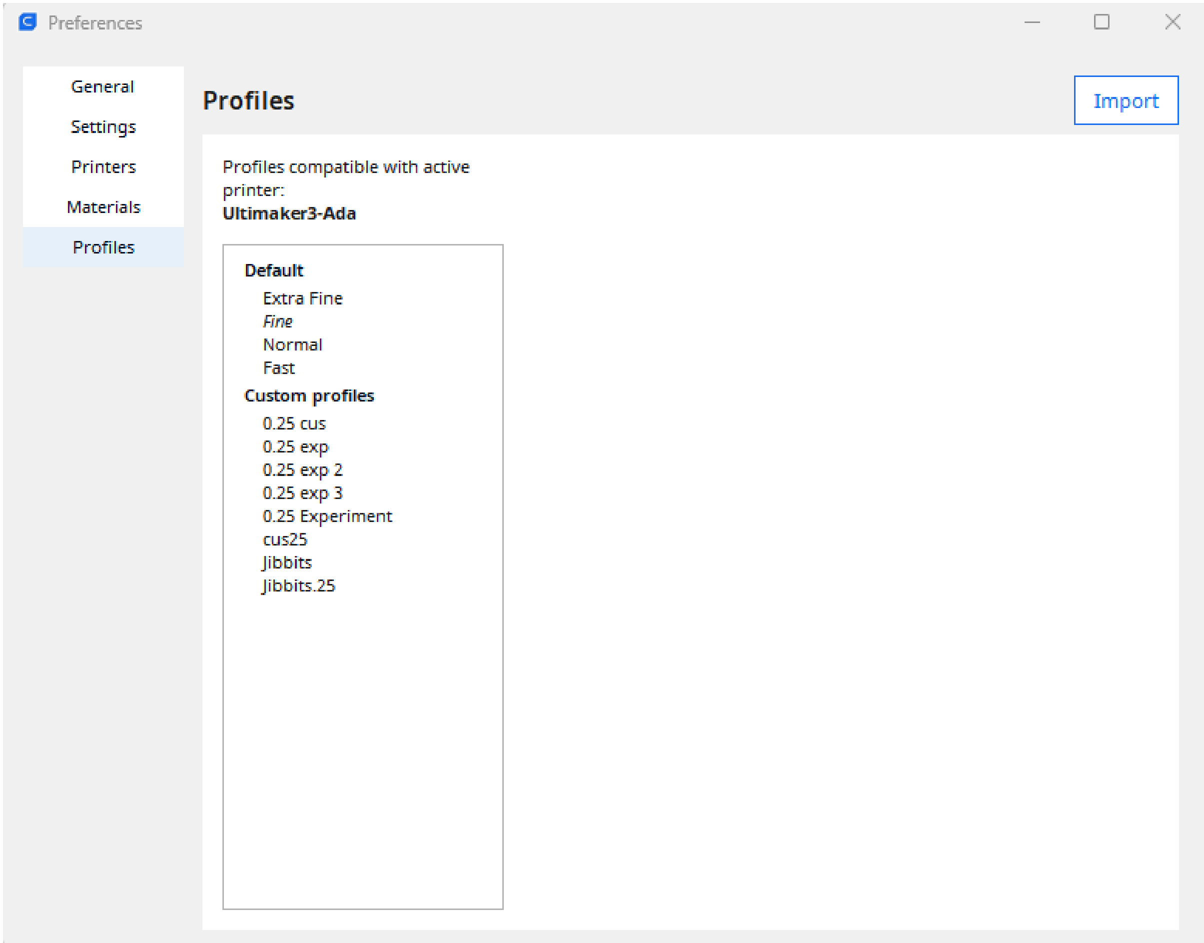


5. Check the scale of the object by clicking the Scale icon. Ensure it's the desired size. To change the size, click on the scale % boxes or the mm boxes.
6. Change the print settings
    a. Click on the toolbar in the top right corner to change the print settings.
    b. Click on the profile drop down menu and select the appropriate profile. This changes the layer height and the print speed. The finer profiles will tend to have a lower speed, which results in more accurate prints.
    c. Some important headings to click on and modify the settings of if necessary are Infill, Support, and Build Plate Adhesion.
7. Once the settings are as desired, click on the blue Slice button on the bottom right of the screen. Then click on Preview. A slider will appear on the right side of the screen. Move the top handle down to show how each layer of the print will be sliced.
8. Save the project by clicking File then Save. Choose a name and location to save it to.
9. To print the file, click on the Print over network button that comes up in place of the Preview button. Select the printer it should be printed to in the pop-up box.



Printing with 0.25 mm Nozzle in Cura

Download the custom profile from here, and save it to your local storage. Open the Cura, from "Settings" tab on top of the window, select "Configure setting visibility". In the opened window select "Profiles" and then "Import". Find the profile "Cus25" you just downloaded and import it to the Cura. Thereafter, you can use the profile for printing with 0.25 nozzles.

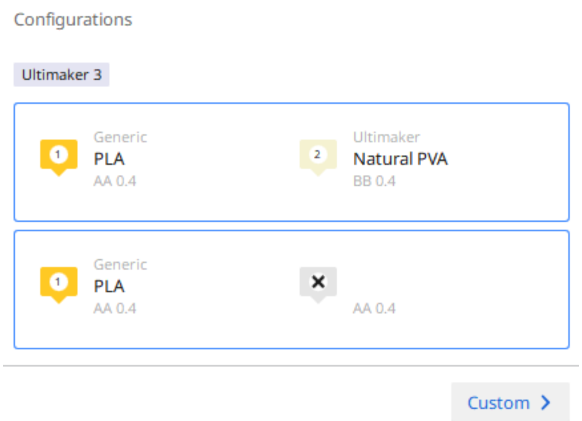


Important things to consider when printing with 0.25 nozzle:

- Using a 0.25 nozzle would not necessarily result in a considerable increase in resolution in Z plane or vertical plane. That is because your layer is not changing. Rather, the most substantial difference would be in horizontal plane or XY resolution.
- Decreased speeds would always help get a better result due to less chance for over shoot.
- Try not to print large models with a 0.25 nozzle as it is not reasonable to do so, miniature models however may need a 0.25 nozzle to maintain a good resolution.
- It is a good practice to keep the line width of the print be equal to the nozzle diameter, that is, printing with a 0.25 nozzle requires a line width of at least 0.25 mm.

Placing Multiple Prints on the Build Plate

Cura will try to fit all of the objects on the build plate but sometimes they're placed off the build plate. If the opened object(s) do not appear then zoom out until they appear. Drag them onto the build plate.



If several copies of the same object are to be printed on one build plate, then:

1. In Cura, add the object to the buildplate
2. Right click on the object
3. Click on Multiply Selected Model
4. Enter the number of additional objects

5. Click OK

PVA Supports

PVA dissolvable supports are used on parts with lots of detail, or parts that need to replicate real-life organic shapes like bone. Please return the PVA roll to a sealed dry bag when the print is complete!

Note: You may desire printing with supports that are mostly PLA, with PVA forming a dissolvable interface where they touch the part. Settings for this method are in Step 5 of the instructions below.

PVA Interface Pros:

- PVA can be very stringy and sometimes unreliable when printed as a tall support
- Reduce the time of the print
- Reduce the cost of the print
- Good for medium-large parts

PVA Interface Cons:

- Removal of such supports may be difficult if they are caught inside the part. Bad for tiny Heroforge D&D miniatures.
- The PVA can take longer to dissolve, as less surface area will be exposed to water.

The thickness of the PVA interface can be increased if you need to remove the supports from an intricate surface and want the PLA to be relatively loose when the PVA is dissolved.



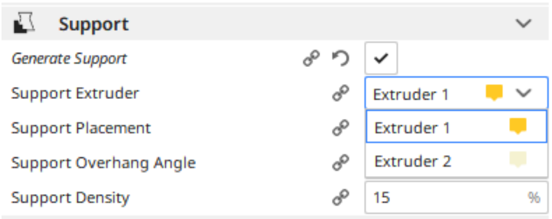
(Before water) (Don't print with pointy ends pointed down!) (after water)




(before water) (after)

This print turned into spaghetti with 100% PVA supports, but removing the partial PLA supports nearly broke its ankles. Could be remedied with a thicker PVA interface.

1. To print with PVA, the printer must be setup properly and be represented accurately in the settings
    a. On the printer, Printcore 1 should be loaded with a AA 0.4 nozzle and be loaded with PLA. Printcore 2 should have a BB 0.4 nozzle and be loaded with PVA.
    b. In Cura, click on the Prepare tab at the top of the screen. Click on the material bar immediately under the Prepare tab and a Configurations window will pop up. If one of the printers is set up using the instructions in step 1.a then select that printer. If neither of the printers can be setup like in step 1.a (due prints currently ongoing, etc.) then click on the Custom button and replicate the settings in step 1.a.
2. In the Print settings menu, click on the Support tab
3. Ensure Generate Support is checked
4. For 100% PVA Supports: Click on the Support Extruder drop-down menu and select Extruder 2 from the list. The default settings for PVA are nearly always sufficient. Skip to Step 6



5. For PLA Supports with PVA touching the part:
    a. Click the stack dropdown menu next to the Search settings bar.
    b. Select Expert or All.
    c. Click on the *Support Interface Extruder* drop-down menu and select Extruder 2 from the list.
    d. Change the Support Interface Thickness to 1.5mm (Check that Floor Thickness and Roof Thickness match)
6. One more consideration when printing with PVA (or any print with two filaments) is adding a prime tower. This is useful for preventing underextrusion, oozing, and feeders grinding the filament. Use a prime tower for prints where not much filament is printed during each layer.
    a. In the Print settings menu, click on the Dual Extrusion tab
    b. Check Enable Prime Tower
    c. If the location of the tower interferes with the print (or you want it closer to reduce printhead travel time) then it can moved by adjusting the X and Y positions

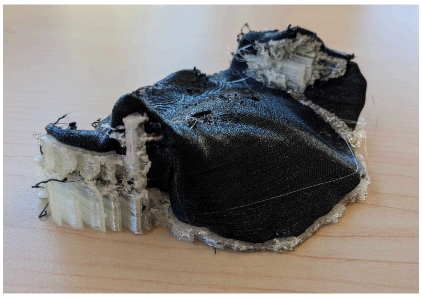
TPU 95a

TPU 95a is a flexible filament so it presents unique challenges compared to PLA. Since the feeder and the nozzle are separated by a significant distance, when the feeder tries to retract the filament it may not succeed in pulling the flexible filament away from the nozzle. This results in a lot of stringing and overall not a clean print if lots of retraction is needed. TPU 95a is best used for objects requiring flexibility that are also continuous and don't require support. Both Ultimaker and generic TPU work well, in both cases it is recommended to increase the temperature (see possible settings for odd filaments) to reduce blockage.

1. To print with TPU 95a, the printer must be setup properly and be represented accurately in the settings. **Since the S5s have filament sensors, it is highly recommended to send jobs requiring TPU95 to these printers.**
    a. On the printer, Printcore 1 should be loaded with an AA 0.4 nozzle and TPU 95a.
    b. In Cura, click on the Prepare tab at the top of the screen. Click on the material bar immediately under the Prepare tab and a Configurations window will pop up. If one

of the printers is set up using the instructions in step 1.a then select that printer. If neither of the printers can be setup like in step 1.a (due prints currently ongoing, etc.) then click on the Custom button and replicate the settings in step 1.a.

2. The default TPU 95a settings provided by Cura work well most of the time. Some print settings that can be adjusted are:
    a. Profile: the layer height and speed will be changed based on the chosen profile. Note: The smallest layer height that works well with TPU 95a is 0.1 mm.
    b. Build plate adhesion: the default brim can be replaced with a skirt if need be. The brim is more difficult to remove with TPU 95a than with PLA.
    c. Speed: 15mm/s
3. Supports
    a. For projects requiring TPU95a and support, it is possible to use both PVA and PLA supports. PVA supports are very difficult to print with and only work well if the supports are directly on the build plate.
        i. For supports that need to be built on an existing part of the model, this type of support is very unreliable and does not produce good results (left of the image vs right part of the image).



    b. PLA supports, however, work very well with TPU95a. To use this type of support, we need to use two AA 0.4 print cores.
        i. The final printed model does require clean-up, yet these are much easier to remove compared to TPU supports and the colour can be matched in order to make any imperfection less visible. In the above photo, supports have been removed from the right part of the model (with the overhang) and leave a clean finish. On the left part of the model, imperfections have not yet been removed, but can easily be cleaned up with a pair of pliers.

For more possible settings:
https://docs.google.com/document/d/1O5HRDP38VRsMucnhgIXIN5IlWPkOx732bZCmRBOw7w8/edit?usp=sharing

PETG

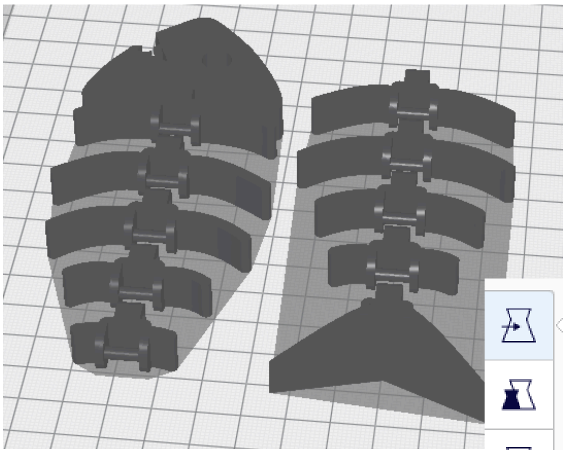
PETG and r-PET are materials that are more heat resistant than PLA, but are a little more stringy. The DSC has both brand-name and generic PETG filament, both work well. When using PETG, make sure to use the PETG settings available on Ultimaker Cura. These settings work well, for all brands of filament we carry, but it can be beneficial to increase slightly the temperature (increase by 5 degrees as needed) to reduce clogs.

Currently, PETG is only offered in white, red, black (PETG) or translucent blue (r-PET) and costs 10c/gram.

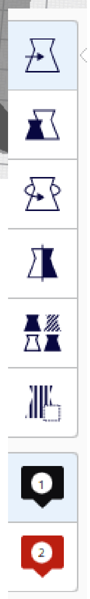
Two-Colour Printing

Printing with two colours of PLA is used when two colours are needed in the same object. It's not requested very often by students but it's fun to know.

1. To print with two colours of PLA, the printer must be setup properly and be represented accurately in the settings
    a. On the printer, both printcores should have an AA 0.4 nozzle and be loaded with PLA.
    b. In Cura, click on the Prepare tab at the top of the screen. Click on the material bar immediately under the Prepare tab and a Configurations window will pop up. If one of the printers is set up using the instructions in step 1.a then select that printer. If neither of the printers can be setup like in step 1.a (due prints currently ongoing, etc.) then click on the Custom button and replicate the settings in step 1.a.
2. For two-colour printing, the object must be imported as two different objects, one for each colour to be printed. When they are initially placed on the buildplate, they probably won't be aligned. Alignment methods will vary depending on the shape of the objects and how they will fit together. Some methods include:



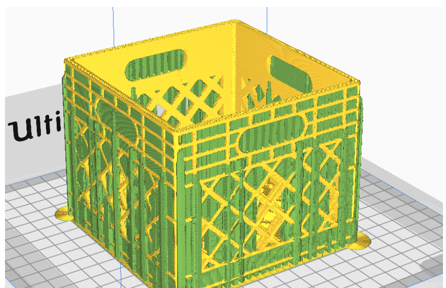
    a. If the centers of both parts should be aligned, then click on the Move tool and enter "0" for all coordinates
    b. If the objects are offset from each other by a known amount, then click on the Move tool and change the coordinates to the known values
    c. If it's a "flexi" object, use the Move tool to drag the objects into an orientation where the connecting pieces do not interfere with each other



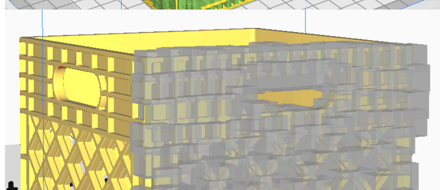
3. By default, both objects will be set to be printed with Extruder 1. Click on the object that will be printed in Extruder 2's colour. On the toolbar on the left of the screen, click on the Print Selected Model with Extruder 2 icon
4. Continue as usual with the rest of the print setup.

Support Blocker (Cura)

The Support Blocker is a function in Cura where you can turn Supports on but then block supports from generating in certain areas. It's a bit awkward to work with, but can be useful. Makerbot Print has a setting that enables you to turn off supports under bridges that will not need supports, but Cura does not have this option. So, an object such as this milk crate would end up having extensive, unnecessary supports.

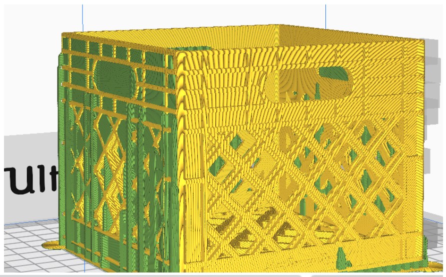


1. First, run a print preview to see where supports will generate and where you might need to prevent supports from building. As you can see to the right, this crate has extensive supports generated,



which would be a waste of material and require a lot of time to clean up.

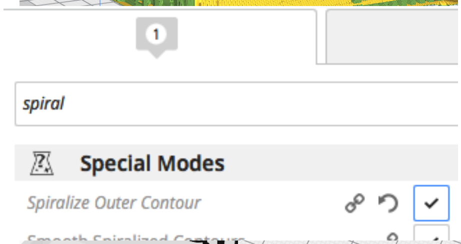
2. Go back into Prepare mode. Click on Support Blocker. (Note: if you have Slice Automatically turned on, you will want to turn it off for this. You can do that in Cura Preferences.) Start clicking on all the areas where you want there to be no supports. Little translucent boxes will appear. These are the support blockers. You only need to put them on the overhangs. They can overlap each other, so when in doubt, you can add lots of them.
3. To remove a blocker, simply click on it again.
4. To the right is an image example showing one side of the crate with blockers placed and then a print preview to show the difference with just one side having blockers applied. There should be supports in the handle of the crate at this scale, so if this object were going to be printed, the blockers would need to be removed at that point of the model.

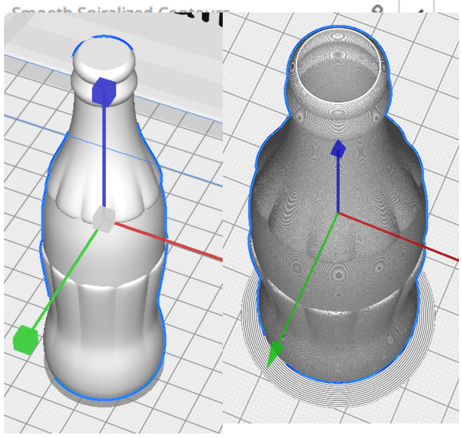


Spiralize/Vase Mode (Cura)

This mode allows you to print just the outside wall of an object to create a fully hollow, thin object. Not something often called for, but more likely to be asked for when someone is looking to create something that they want light to be able to get through or when trying to get as clear as possible when using translucent material (such as when printing a bottle), especially if the model was not designed hollow. Note: this will not create a strong model as it will only create one wall.

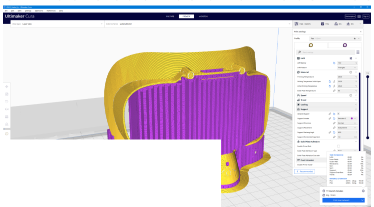
1. Insert model
2. Search for "spiral" in the print settings
3. Check Spiralize Outer Contour. This should auto-select Smooth Spiralized Contours
4. Run slice and check Preview. You can see from the before and after to the right that the model is now hollow.

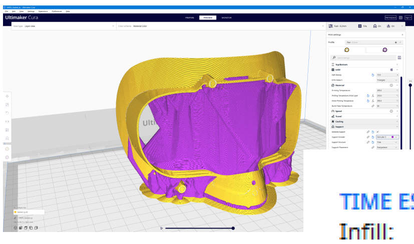




Tree Supports (Cura)

Cura has an alternate type of support called tree supports that can greatly reduce the time and support material use of some prints while still providing adequate support. Rather than printing uniform scaffolding, supports are printed as hollow tubes with fractal-like arms or branches giving them the appearance of trees. An ideal situation to use tree supports is a print consisting of a thin convex shape with a large gap in the middle such as a full face mask.





TIME ESTIMATION

Infill:	00:00	0%
Inner Walls:	00:32	5%
Outer Wall:	00:47	7%
Retractions:	02:04	19%
Skin:	00:18	3%
Skirt:	00:22	3%
Support:	05:45	52%
Support Interface:	00:49	7%
Travel:	00:24	4%

MATERIAL ESTIMATION

PLA	3.57m	28.2g	€ 0.00
PVA	6.99m	54.9g	€ 0.00

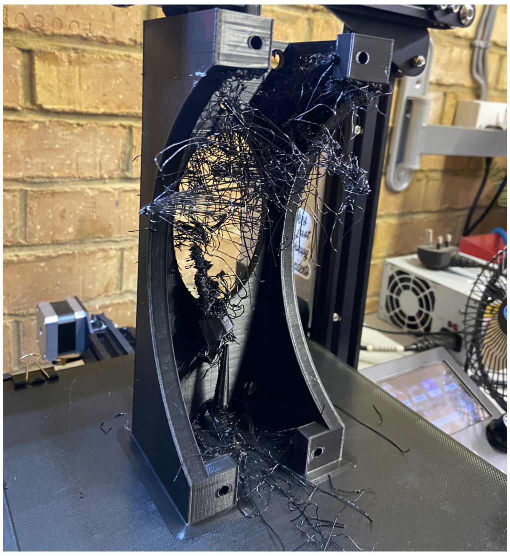
TIME ESTIMATION

Infill:	00:00	0%
Inner Walls:	00:32	8%
Outer Wall:	00:47	12%
Retractions:	01:29	23%
Skin:	00:18	5%
Skirt:	00:23	6%
Support:	02:20	36%
Support Interface:	00:09	3%
Travel:	00:23	6%

MATERIAL ESTIMATION

PLA	3.55m	28.1g	€ 0.00
PVA	2.85m	22.3g	€ 0.00

However, tree supports do not always offer an advantage as seen in the next example.



Things to consider while using tree supports:

a. In case of low computing power, it may be harder for Cura to slice complex models with tree supports a it requires a lot of calculation in contrast to normal supports which are vertical pillars.
b. On models with a large flat surface the normal support may still be a better option.
c. <u>Do not use tree supports when printing PVA</u>. PVA is susceptible to failure as it is sensitive to humidity, which most often would result in underextrusion. Combined with hollow trunks in tree supports, it would not result in a perfect support.
d. Brittle filaments (e.g., PVA) are more likely to fail when using tree supports with thin walls as they may break easily during travel.
e. With prints that require high tree branches for printing, increasing brim or infill density for the supports to 10% would be helpful as it woul prevent the trees brom breaking or falling.

To enable tree supports:

1. Navigate to supports in print settings and click on the three sliders icon to view the setting visibility.
2. Under support settings, check “Support Structure”.
3. With the support structure settings now listed, click the drop down menu and select “Tree”.
4. Upon slicing, the print will now use tree supports.

Per Model Settings (Cura)

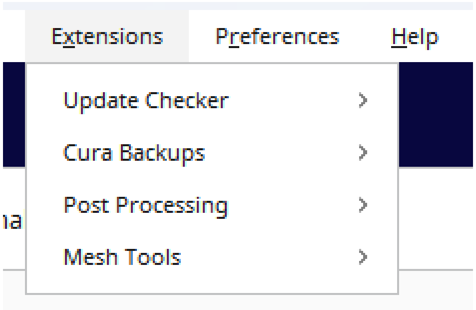
In some cases there will be many small prints in the que that require the same material and layer height, but not the same infill density or choice of dual extrusion for supports. By using Cura's per model settings feature, it is possible to print these parts at the same time.

1. Prepare the global print settings as usual.
2. Select a part that requires different print settings.
3. Click the "Per Model Settings" button on the left side of the Cura window. It has an icon consisting of four differently shaded squares. Then click "Select Settings".
4. Check any settings that need to be changed in the popup window that opens, then change their values in the per model settings tab to the left.
5. Repeat steps 2-4 for any other parts requiring different settings from the rest of the print.

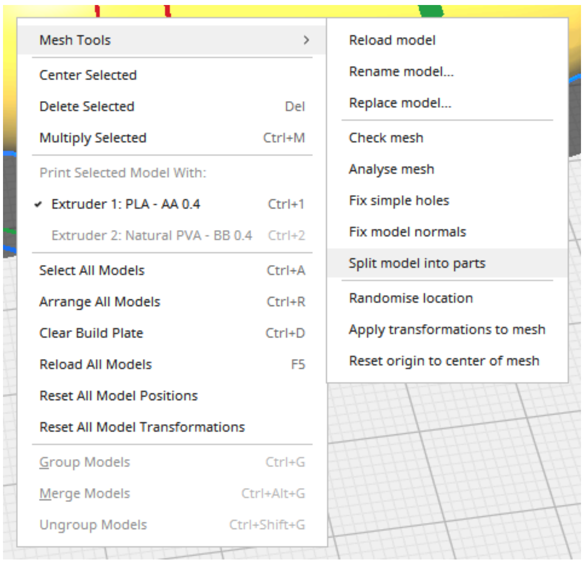
Split Multi-Part Models (Cura)

On rare occasions, a user will submit a 3D model made of separate parts that are arranged too awkwardly to print. There is a plugin that allows these to be split into independently moveable objects.

1. Download the latest release of the Cura plugin [Mesh Tools] if it is not already installed under the "Extensions" tab.



2. Drag the downloaded file into the Cura window.
3. Right click the multi-part object, go to "Mesh Tools", and click "Split model into parts".



3 Common Printing Issues

In this section, a few of the common issues experienced at the DSC will be discussed. A google search for any of these problems will come up with a ton of results and some of them go really in-depth. This section is just a quick guide to common issues, it is in no way a complete list.

Print Lifting Off the Build Plate

Objects will sometimes peel up at the corners while printing.

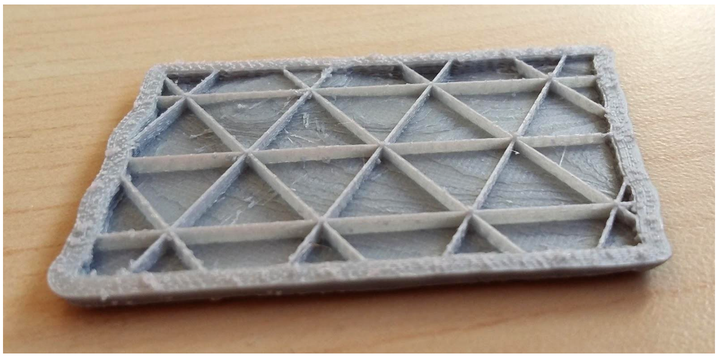
Since the Ultimakers have heated build plates and the DSC only prints with easy filaments this isn't much of an issue. A thin layer of hairspray is used to help with build plate adhesion so if a print isn't sticking it's probably time to reapply. When lifting does occur, it's typically when printing a much larger object than usual, where at the edges there isn't as much hairspray.

Underextrusion

Underextrusion is when filament is being printed but less than the expected volume flow. It can be seen most easily as holes in the normally solid walls of the infill pattern. The example to the right shows a print where the temperature was set too low and had to be restarted.

There are several reasons why underextrusion would happen on an Ultimaker. The first is the temperature being set too low. The PLA at the DSC prints best at 200-205 degrees celsius. It's possible for the print temperature to accidentally be set lower in Cura. The second is there being a partial clog of the nozzle. With the AA 0.4 nozzles, they do not require cleaning very often because printing PLA doesn't produce a lot of buildup inside the nozzle. The BB 0.4 nozzles on the other hand get clogged often enough because the PVA will carbonize inside the nozzle. One way to avoid this is to unload the PVA filament when it's not in use because even when only printing PLA, the PVA nozzle will still be heated up to about 40 degrees, causing carbonation. Underextrusion in both nozzle types can be fixed by doing a series of "hot pulls" and "cold pulls". This can be accessed on the printers by going into the "Maintenance" section on the user interface.

No Extrusion



Sometimes chronic underextrusion will lead to a complete stop in filament extrusion, or extrusion could stop all of a sudden. To the right is an example of an "all of a sudden" no extrusion case.

The simplest cause to this problem is there's no more filament on the spool. The solution is to restart the print with

a new spool of filament because unfortunately the Ultimaker 3 printers don't pause the print when they run out of filament (however, the Ultimaker S5 does pause).
Another cause is that the filament is stuck on the way to the feeder which the feeder cannot overcome with its strength. This can happen if there are two filaments on the spool holder and the filament guide isn't used. Also, it can happen when the windings of the spool get tangled. The solution to these problems is varied, for example putting on the filament guide or untangling the spool.
Sometimes a partial clog can turn into a full clog and prevent any filament from extruding. The fix is the same as with underextrusion.

To unclog a print core, follow the instructions found in this video including doing a hot pull, a cold pull, and as a last resort, using the cleaning needles.

4 Maintenance Procedures

XY Calibration

To ensure the accuracy of material placement with dual extrusion, you should run an XY calibration whenever a print core is changed. You should be prompted to run an XY calibration after loading a new print core, however you can run one any time by going to SYSTEM -> Maintenance -> Calibration -> Calibrate XY Offset
The printer will print single layer calibration strips for the X and Y axes for the next ~20 minutes. After the print is complete, carefully remove the glass build plate and place it over one of the calibration sheets found on the shelf at the back of the 3D printing / storage room. Choose an offset that lines up with the most collinear pair of opposite extruder lines.
Ultimaker XY Calibration Guide

Play on Axles

* If you notice some play on only one of the axles, you can try to fix it by following the steps below. However, if the play is too much, or more than one axle is affected, you should realign the axles instead (see Axle Alignment). Do not over-tighten the pulleys, as this can make the print head shafts not perpendicular to each other, and affect the print quality.
The pulleys are responsible for locking the X and Y axles in their positions. However, over time, some of the pulleys may become slightly loose due to wear and tear or other factors. This can affect the alignment of the axles and cause them to have some play or movement. This can lead to problems with the print quality, such as inaccurate dimensions, poor surface finish, or layer shifting. It is recommended to check for play on the axles at least **once every three months**.
In case of **minor** play:

- First check which of the axles has play.
- Loosen the pulleys on that axle. Note that the set screws of the pulleys are not always easily reachable. Adjust the position of the print head until you can reach the set screws. Loosen them, but do not remove them from the pulleys completely.
- Correctly position the axle:
- The X axles should be positioned in the middle.
- The Y axles should be pushed toward the front of the printer.

- While ensuring that the axle stays in place, firmly push the pulleys toward the frame.
- Tighten the set screws. Make sure the pulleys are tightened securely. If they are too loose, this can lead to print quality issues such as layer shifting.

Axle Alignment

To print accurately, the print head shafts need to be perpendicular to the X and Y axles in the frame. This means that they form a 90º angle with each other. If the print head shafts are not aligned properly, this can distort the dimensions and shape of your print. You should calibrate the printhead and align its axles if you notice this problem, or if you have considerable play on the axles.
You will need the following tools:

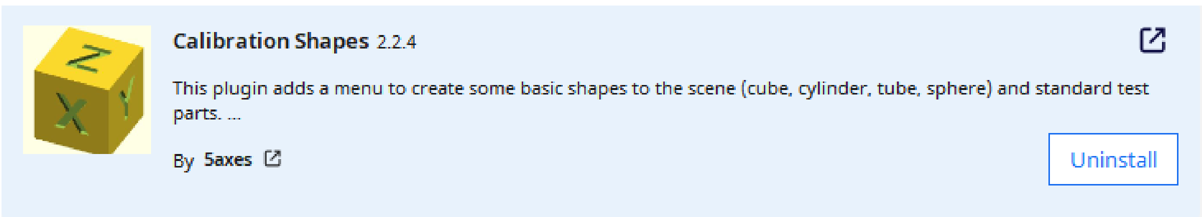
- A 2.0 hex screwdriver or a ball-head screwdriver
- A 0.5 Nm torque screwdriver (optional, but recommended)
- Two calibration sticks (one for the X axle and one for the Y axle)
- Depending on your printer model, you will need different sizes of calibration sticks for the X and Y axles. For the Ultimaker S3 and S5, you will need an 8mm calibration stick for the X axle and a 10 mm calibration stick for the Y axle. For the Ultimaker 2 family and the Ultimaker 3, you can use the same calibration sticks for both axles.

Find the **complete guide** here, follow the procedure step by step as explained. A good case scenario could be found in Ada Troubleshoot documentation in here.

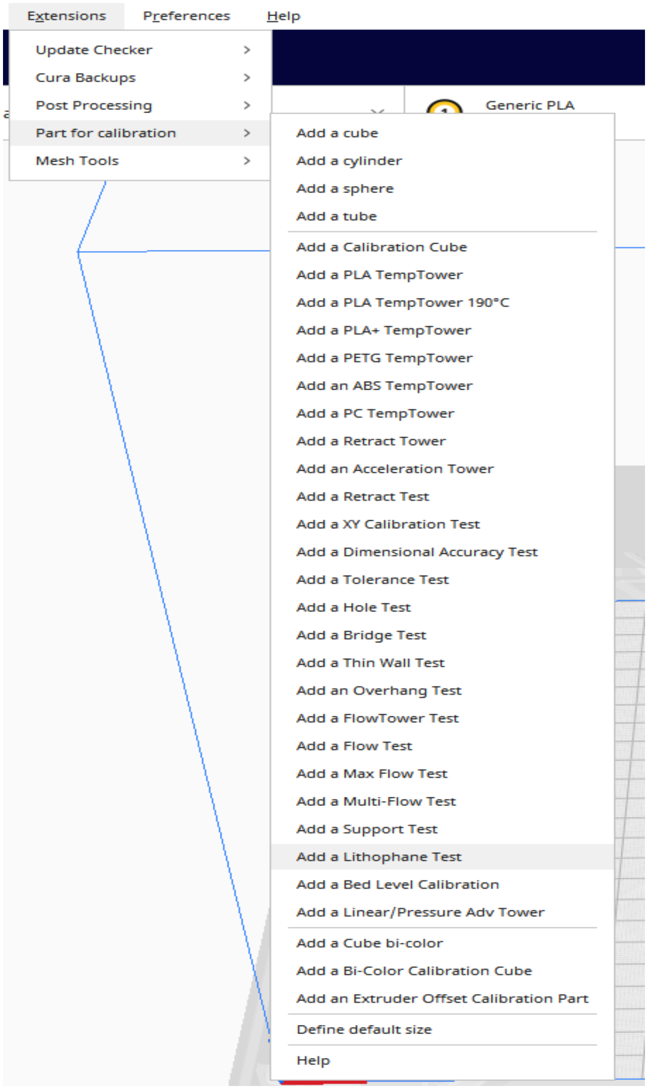
Bed Level Issues

If gaps were present between lines (as could be seen in the first layer), then the printer needs maintenance. There is a good chance that consistent gaps could be due to underextrusion, however, if you are sure that underextrusion is not the problem or the gaps are inconsistent, then another possibility is that the bed is not correctly levelled. It is important to make sure that the bed is level every once in a while. To understand the current status of the bed level, follow the steps below;

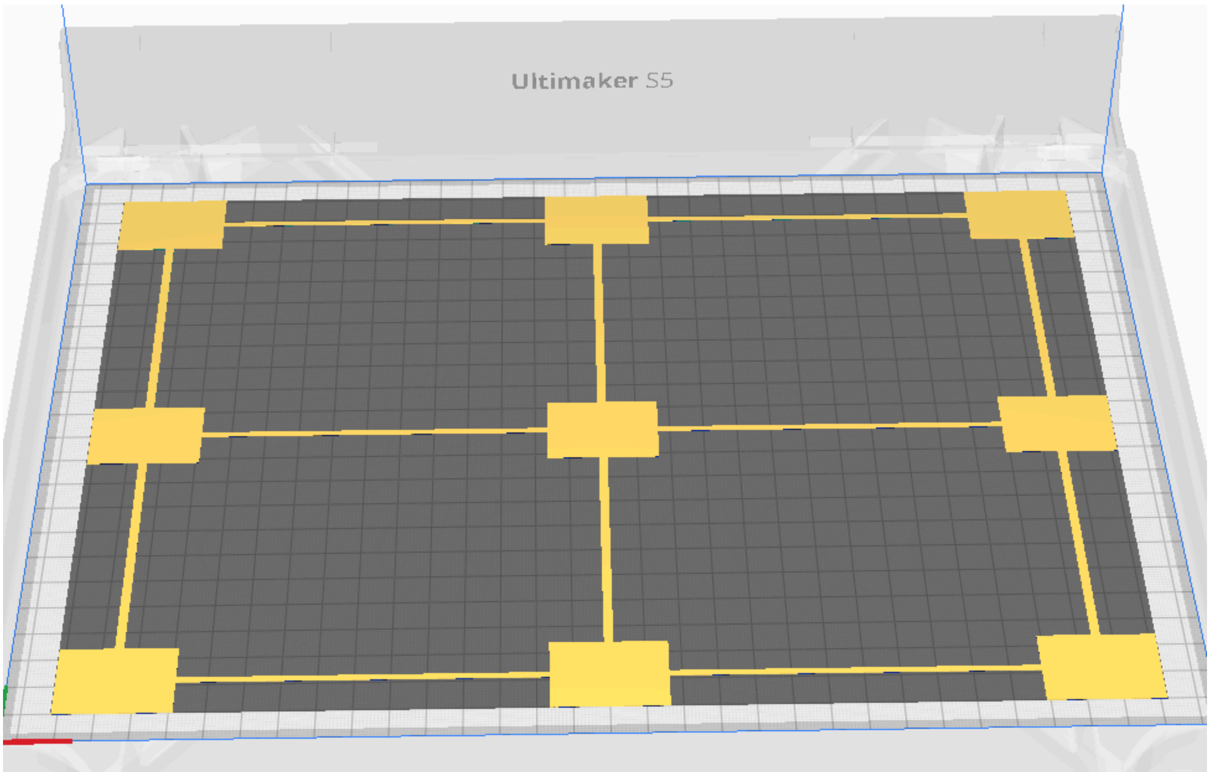
1. Make sure you have the plugin/extension “Calibration Shapes” by Saxes installed on Cura.



2. Open a new empty project, head to “Extensions” -> “Parts for calibration” -> “Add a bed level calibration”.



3. Notice that a calibration model is added on your print bed, and that this model would be different depending on the printer size.



4. Change the print setting to the following setting and print the model:
   - Nozzle Size : 0.4
   - Initial Layer Height : 0.2
   - Line Width : 0.4
   - Wall Line Count : 3
   - Fill Gaps Between Walls (Only for release of Cura previous to V5) : Nowhere
   - Build Plate Adhesion Type : None
   - Initial Layer Speed : 20 mm/s
   - Outer Before Inner Walls: True
5. After printing is done, check for gaps between lines and find out which sides of the bed need to be lifted in order to eliminate the issue. You can increase/decrease the bed height by slightly playing with screws below the bed.
6. If the issue still exists or there are multiple regions with gaps, consider doing a complete manual bed levelling from the maintenance settings of the printer.

5 Examples of Prints

The examples presented are to give a sense for the different settings and why certain settings are used in specific cases. Any of these items that were printed on a Makerbot could also have been printed on an Ultimaker, unless otherwise specified.

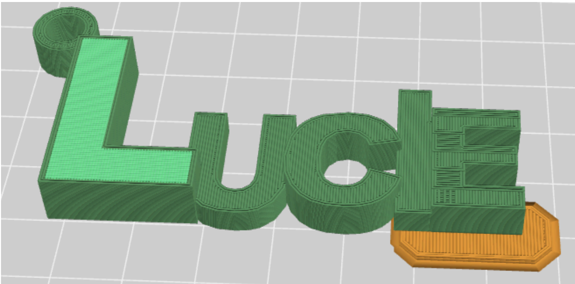
Workshop Activities

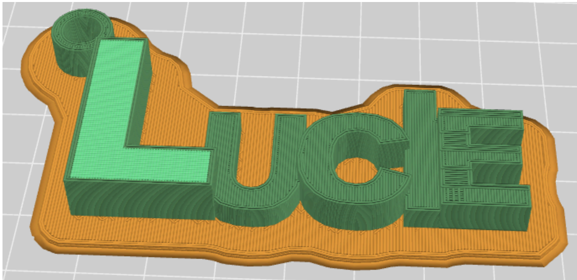
The workshop activities (particularly the Keychain activity) are some of the most common things printed at the DSC.

Keychain

Printer Used: Makerbot
Settings Profile: Balanced (0.2 mm layer height)

Infill: 20% (default)
Support: PLA
Special Settings: None

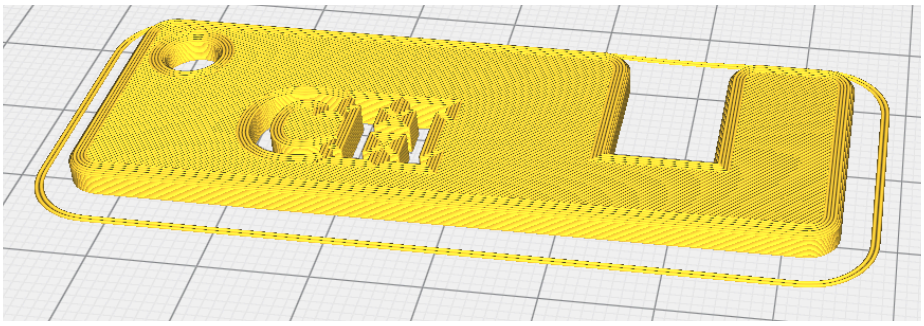




This is usually the simplest, easiest print. Since this is often the very first 3D object someone designs, it’s worth thoroughly looking it over. The most common problem is that the letters don’t touch enough to print as a strong object. Turn supports on because some of the letters are often placed higher than the others by mistake and Makerbot won’t print a raft under those areas (top left picture). With supports turned on then a raft is printed under the whole object (top right picture).

Phone Stand Keychain

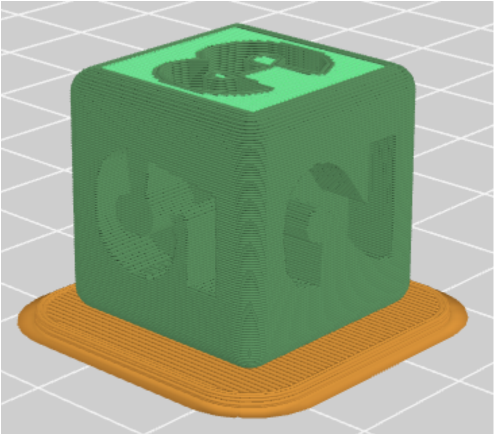
Printer Used: Ultimaker
Settings Profile: Normal (0.15mm layer height)
Infill: 20% (default)
Support: None
Special Settings: Used a skirt instead of a brim

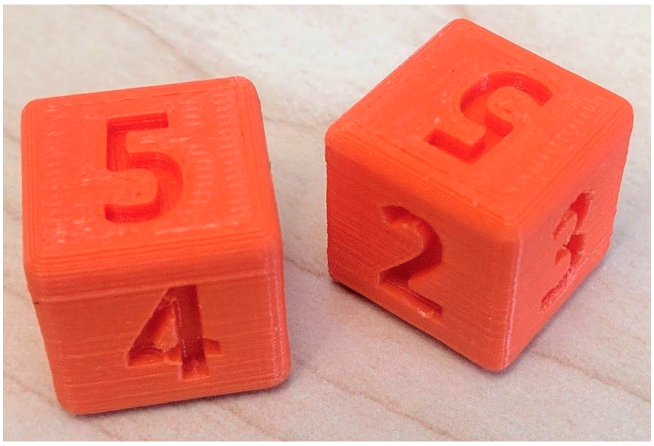


This would print fine on the Makerbots but the heated build plate of the Ultimakers makes for a nice finish on this part. The Normal profile takes a little longer than the Fast profile but gives the part a “smooth” finish. Infill doesn’t matter. This part has a lot of surface area on the build plate so it will stick just fine without a brim but a skirt is needed to prime the nozzle well. The skirt requires no post-processing of the part compared to a brim, which can be difficult to remove cleanly.

Dice

Printer Used: Makerbot
Settings Profile: Balanced (0.2 mm layer height)
Infill: 10% (default)
Support: None
Special Settings: None





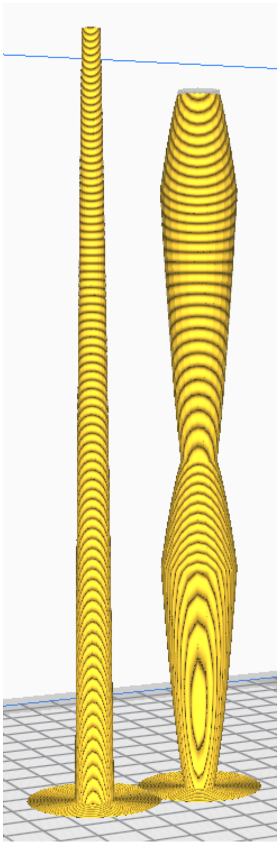
If support is enabled then there will be support material printed in the letters which is difficult to remove. The minor deformation from not having support looks better than scratch marks from removing support.

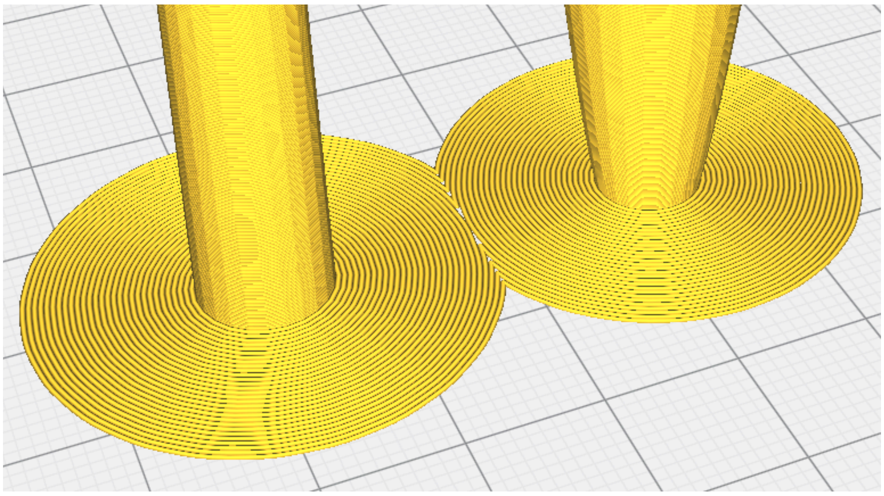
Chess Pawn

Printer Used: Makerbot
Settings Profile: Balanced (0.2 mm layer height)
Infill: 10% (default)
Support: PLA
Special Settings: None

Wand

Printer Used: Ultimaker
Settings Profile: Normal (0.15mm layer height)
Infill: 20% (default)
Support: None
Special Settings: Extra wide brim and slowing down the print speed halfway through the print



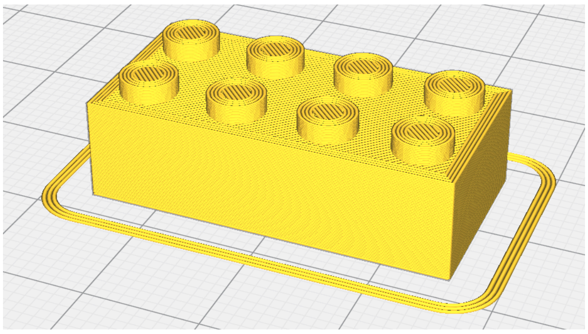


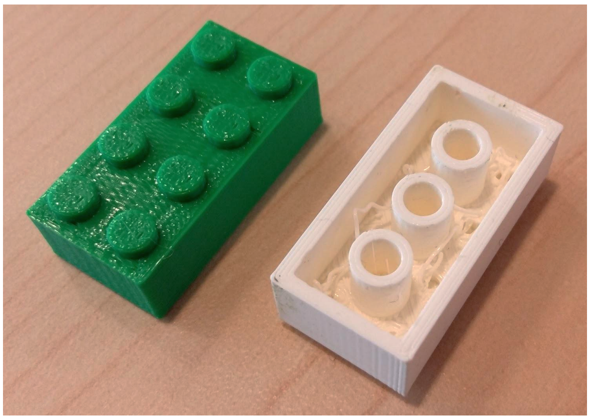
This print can be finicky because the printhead has a tendency to knock over one of the wand halves when rapidly moving between them. Make sure the student has sent the wand as separate files because it may be more reliable to print them one at a time. The brim is 12mm wide to prevent the tall objects from lifting off the build plate when the print head moves between the parts. On the

Ultimakers the print settings can be edited as it prints so the speed is lowered to 70% of the original speed about halfway through the print to reduce the chances of the parts lifting off the bed.

Lego

Printer Used: Ultimaker
Settings Profile: Normal (0.15mm layer height)
Infill: 20% (default)
Support: None
Special Settings: Used a skirt instead of a brim and turned the speed down to 60 mm/s





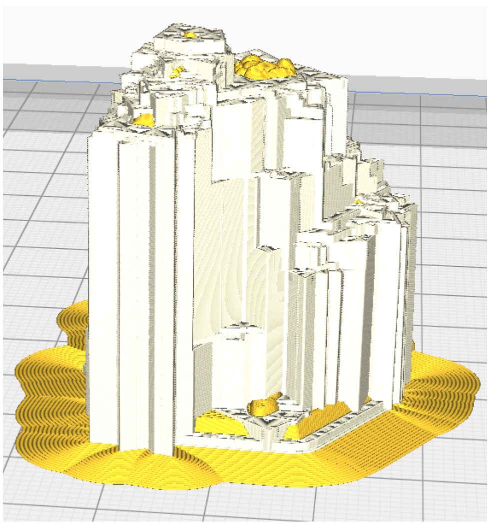
The lego should really be printed with supports but it would be too difficult to remove from the interior so a poor-looking inside is the compromise. If true dimensions are desired, slow down the normal profile to 60 mm/s. The Makerbots with default settings can also be used if the dimensions don’t matter too much.

Non-Workshop Print Examples

These examples are provided to give a broader scope of the types of print jobs the DSC handles.

Dungeons and Dragons Miniature

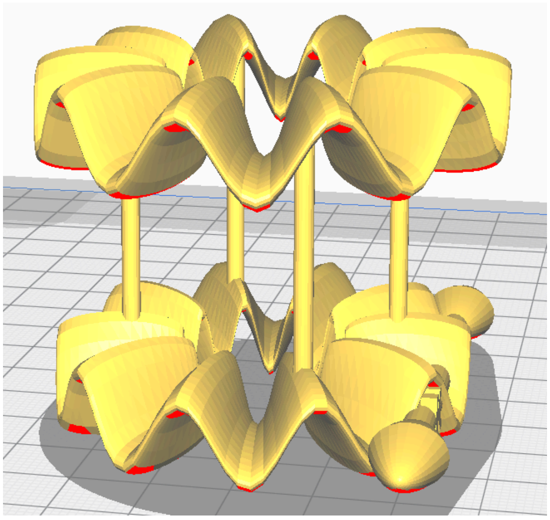
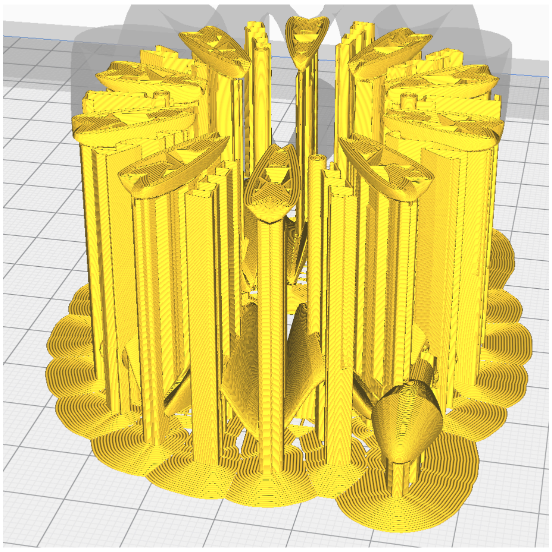
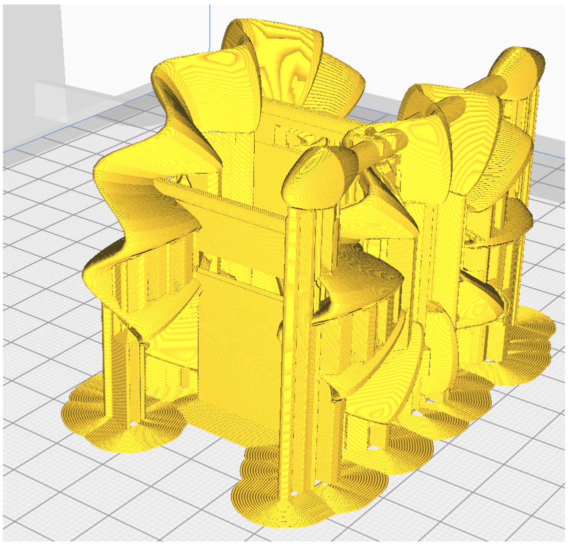
Printer Used: Ultimaker
Settings Profile: Extra Fine (0.06mm layer height)
Infill: 20% (default)
Support: PVA
Special Settings: Enable Prime Tower



Miniature models for Dungeons and Dragons, or other role-playing games, are fairly popular. The files usually come from a website called [Heroforge]() as pictured on the left. These files need to be printed on an Ultimaker because of the fine detail and PVA support (middle picture). If printed at the scale that Heroforge defaults to, not all of the detail will be visible (right picture) but it still looks pretty good. If there are several figures being printed at once then a prime tower is not needed.

Weird-Shaped Thing

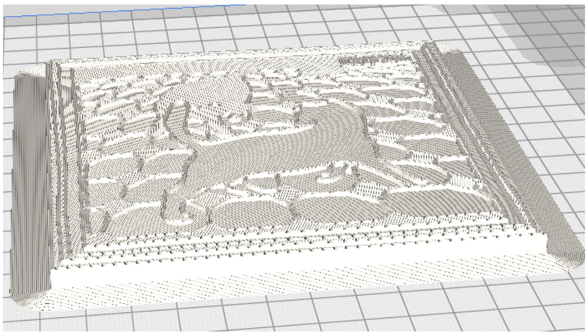
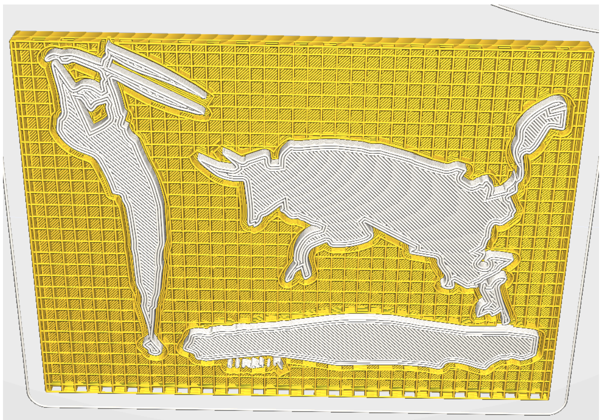
Printer Used: Ultimaker
Settings Profile: Fine (0.1mm layer height)
Infill: 10%
Support: PLA
Special Settings: None

The student requested a fine layer height because this was a part of an art project, so they didn't want to see the 3D printing lines in the model. Otherwise, a larger layer height would have been fine. The default orientation (top left) initially seemed good for this object, until scrolling through the object showed the bottom of the ripples being printed on spindly supports. This would normally be ok but since the student wanted a good surface finish the orientation was changed (bottom). One might ask: if they wanted a good surface finish why not use PVA support? The answer is to save money, sandpaper is cheaper than PVA.

Print Blocks

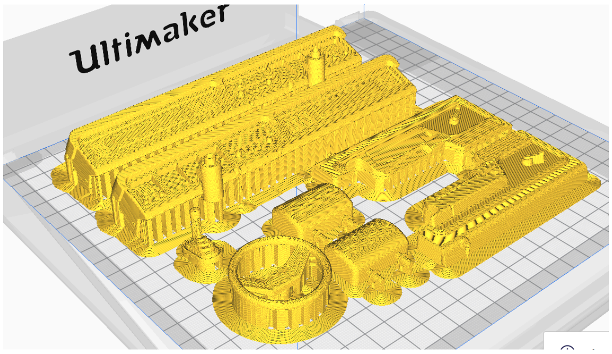
Printer Used: Ultimaker
Settings Profile: Normal (0.15mm layer height)
Infill: 10%
Support: PLA
Special Settings: Various

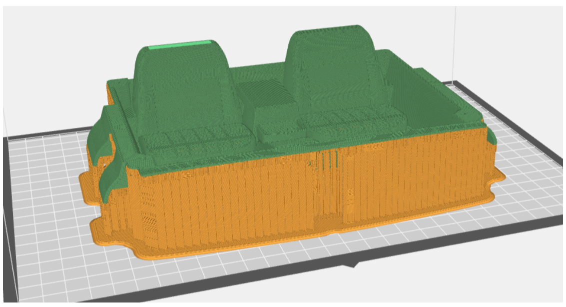



Print blocks are a part of Tiffany's art history workshop. The settings for each print are different since each design presents unique challenges. If the design isn't very detailed, it can be printed entirely in TPU 95a with the design facing up (left picture). If the design is more detailed it may be better to print it upside down in two filaments; TPU 95a for the design and PLA for the base and support (right picture). The only consistent setting is using the Normal layer height but reducing the speed to whatever the fine setting's speed is.

RC Bronco Prints

Printer Used: Ultimaker/Makerbot
Settings Profile: Fine (0.1mm layer height)
Infill: 10%
Support: PLA
Special Settings: None

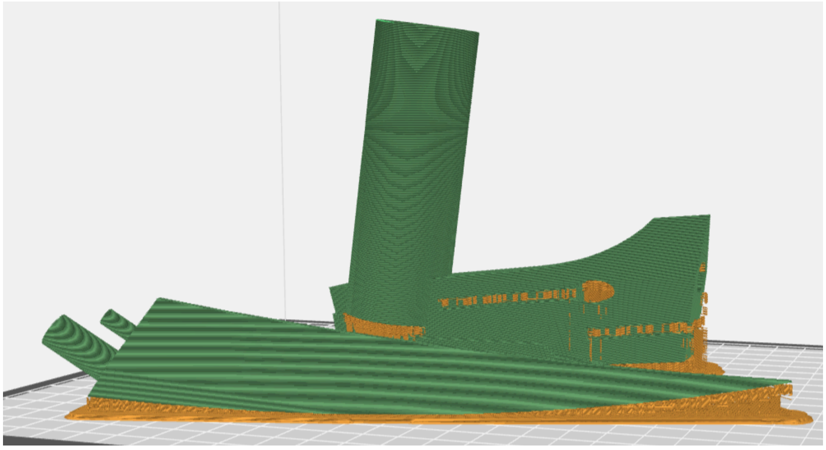




Most of the objects in this print could be made on the Ultimakers without issue (left picture). They had fine detail but only needed one side to look good since they were going in an RC car so PLA supports were sufficient. There was one piece that was too large to fit on the Ultimaker build plate so it needed to be made on the Makerbot (right picture). It would have been preferable to have the student split the piece in half so it could be printed on an Ultimaker but this student had no experience with 3D design and was unwilling to try. The print-lifting-off-the-build-plate issue was explained to them and they agreed to take the risk. It turned out well because on all the edges of the print there was a large amount of support material so the lifting only affected the supports and not the actual object.

Scythe

Printer Used: Makerbot (Replicator+)
Settings Profile: Balanced (0.2 mm layer height)
Infill: 10% (default)
Support: PLA
Special Settings: None



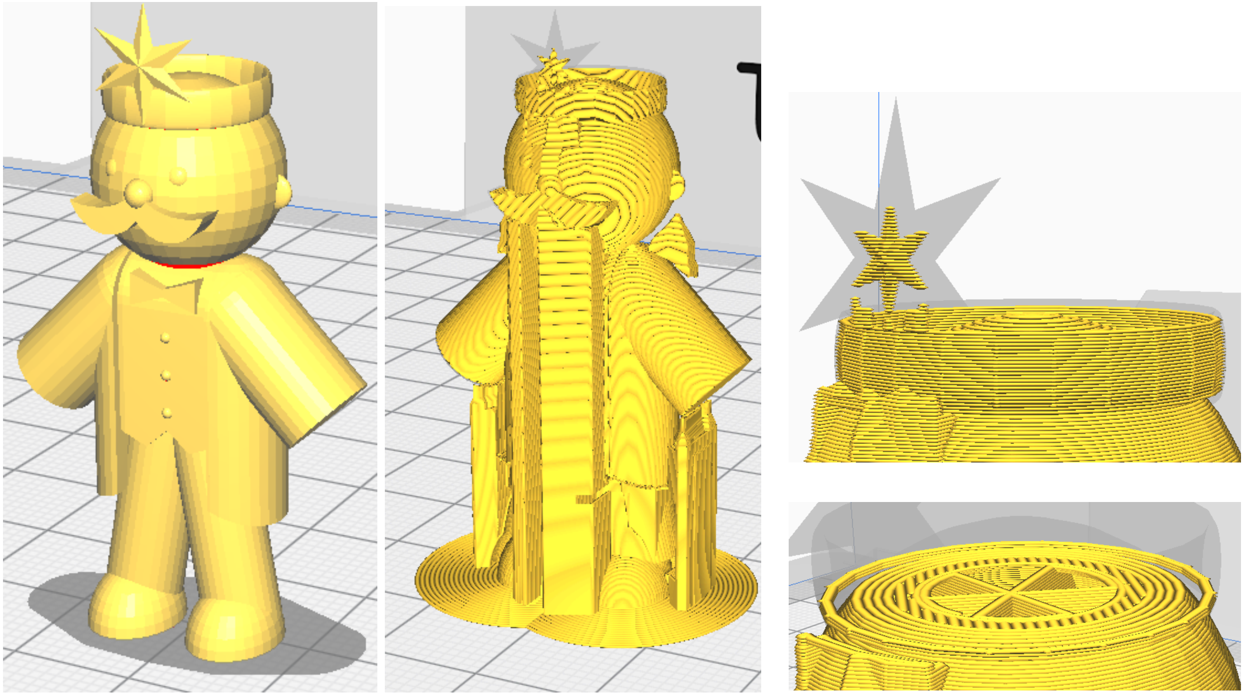
This is a good example of larger objects that might be printed on the Makerbots to prevent unnecessarily making the Ultimaker queue longer. Four objects join together to make a scythe that can be attached to a handle of a wood dowel. Because it's mainly for form, the print doesn't need to be precise, making it perfect for the Makerbots. The Replicator+ was chosen because the objects were too long for the Replicator 5th Gen. The print did lift a little at the ends but since all the objects are curved, there was support material printed in those areas so by the time it was printing the real object it had self-corrected for the lifting.

Problematic Print Requests

These are print requests that either got denied or that were printed but had problems with them.

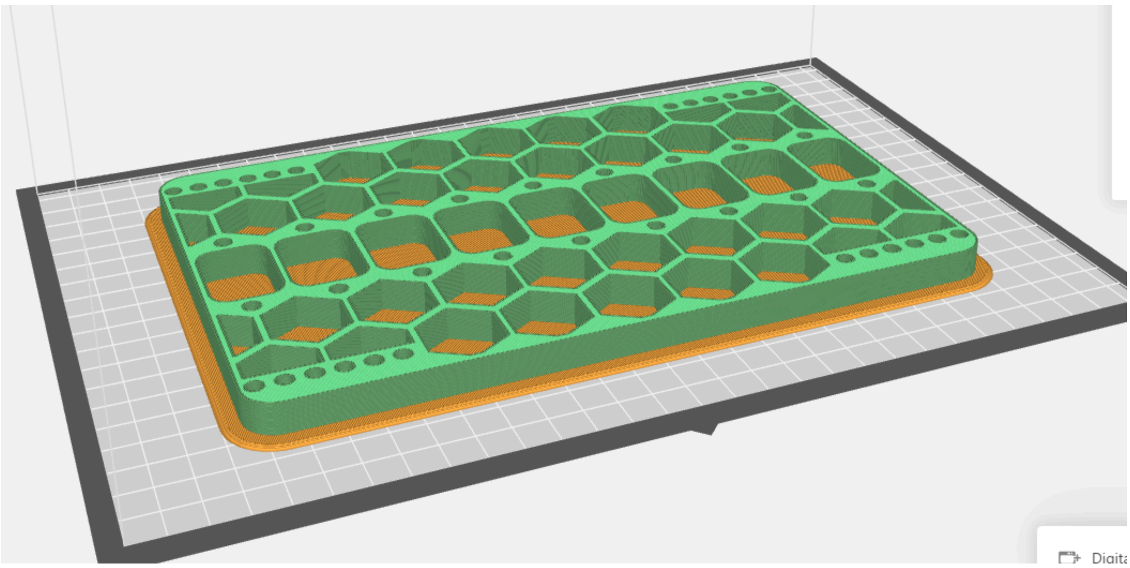
Thin Detail

This print was requested to be made 5 cm tall using PLA supports. The PLA supports would be somewhat inconvenient to remove but the main issue is the headband with the star. The headband itself is not connected to the head, so it wouldn't be attached well when printing. Also the star is too thin to be printed and the small amount that would is not connected to the headband. You can see in the print preview that the outer edges of the star are greyed out, indicating that it will not be fully printed. Dragging through the print preview layers showed the lack of connection between the crown and the head (lower image). The student was asked to increase the thickness of the headband and star but they never replied.



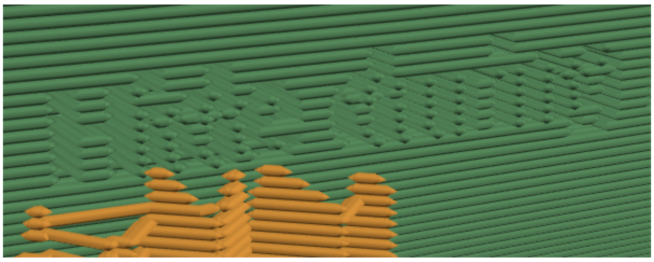
License Plate Standoff

This is an object that should be printed on an Ultimaker but had to be printed on the Replicator+ because it was too long. The student specified that they didn’t want the object split into two pieces. It was printed on the Replicator+ and one corner lifted off significantly from the build plate. It should have been made more clear to the student that lifting was definitely going to happen if it wasn’t split into two pieces.



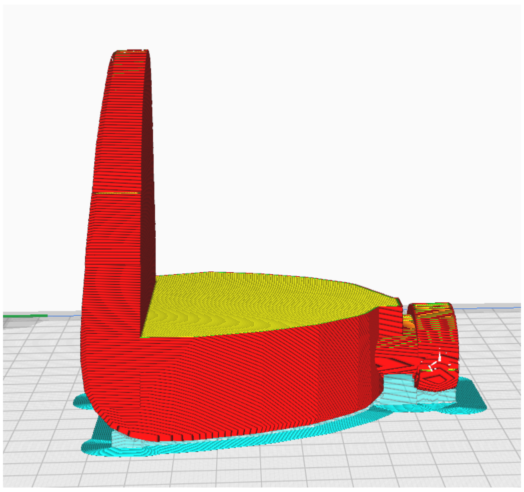
Small Text

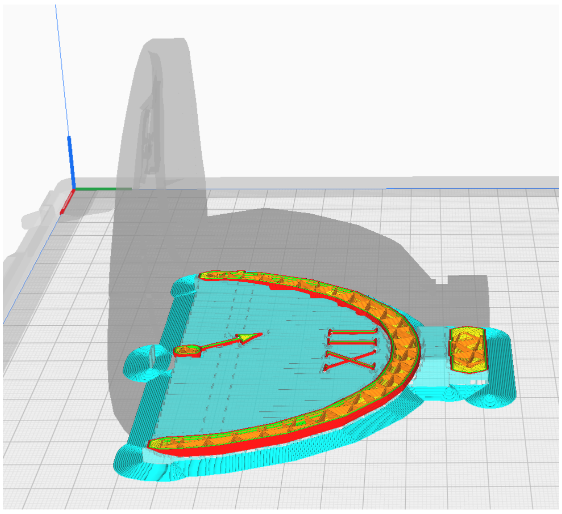
Text is common in TinkerCAD objects but when designing it’s difficult to judge how big the text needs to be in print well. This is an example of an object that looked like the text was on the small side but it might still print ok. It did not turn out well (top-right picture) due to the Makerbot not being able to keep up with all the small radii in the text. There was larger text on the other side of the object that turned out well (bottom-right). The student changed their design to have larger text and it was printed on an Ultimaker. The object turned out quite well after that but unfortunately there is no picture of it.



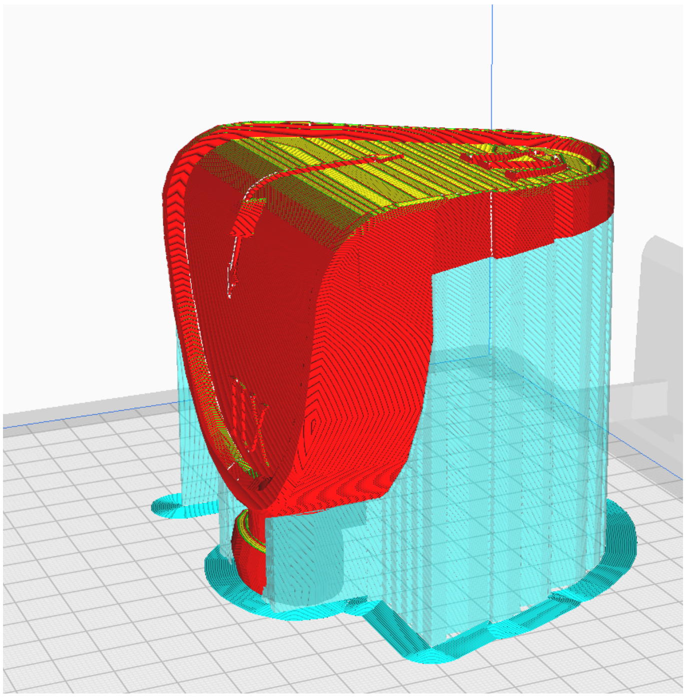
Melting Clock

This is a great example of a non-intuitive orientation for printing. The L-shape combined with detailed features on the outside mean the ‘obvious’ orientation (face-down) won’t work well.

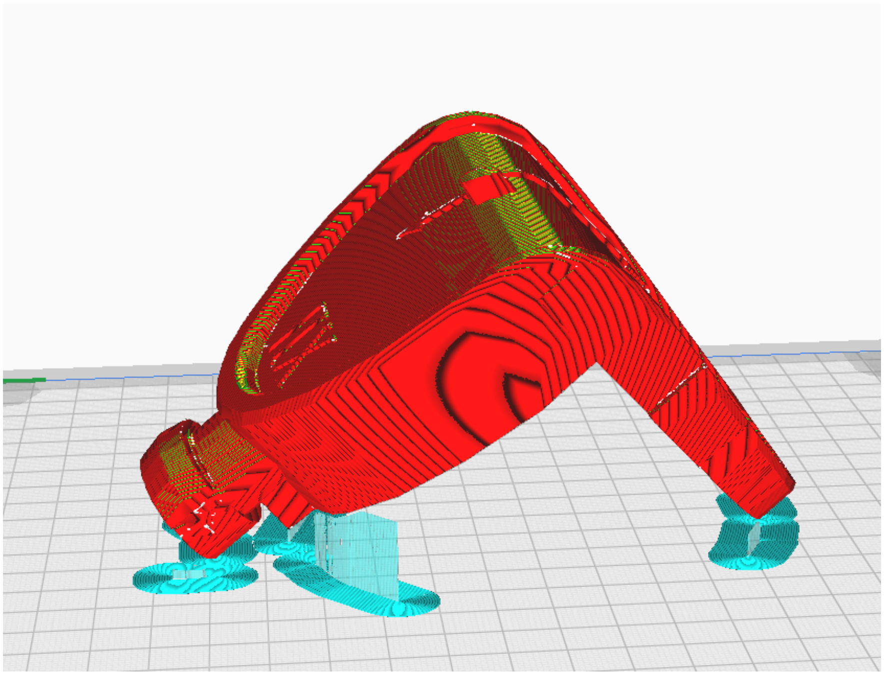




Seems ok at first, but you can see the large area of support required underneath - this would be a nightmare to remove, the clock face won’t look very nice and it wastes material.



This orientation preserves the fine clock face features, but uses too much support.



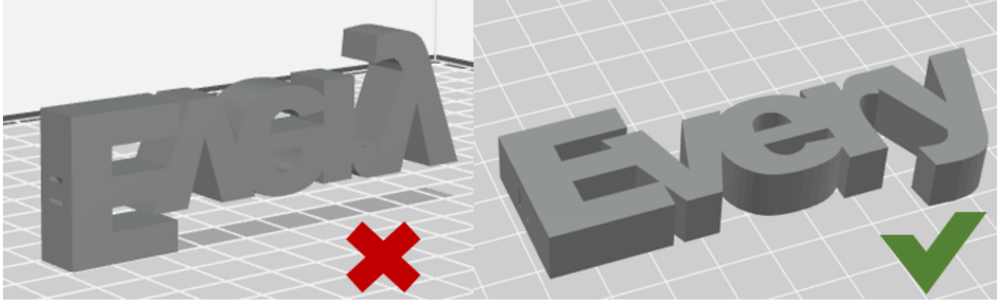
By printing the clock this way, we use minimal support and keep the clock face turned upwards for best results. The tilted angle of the faces are gradual enough to not need support underneath.

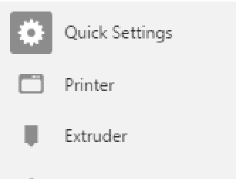
6. MakerBot - Everything you need to know

Setting up a print job

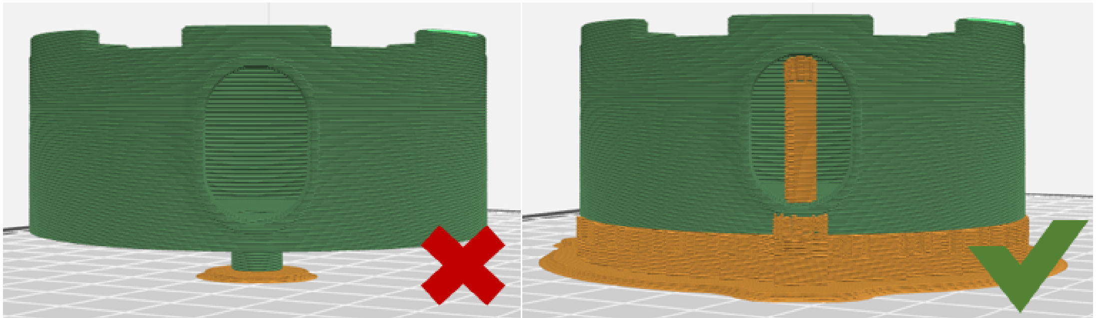
Makerbot Print is used to set up prints for the Replicator+.

1. Open Makerbot Print
2. At the bottom right of the screen, click on the Printers bar. Select which printer this print will be executed on. Some things to consider are print time, print size, and the currently loaded filament colour. The Replicator+ prints quicker and has a longer print bed.
3. In the top left corner, click on the folder icon . Click on the Add Models icon . Select the file(s) you would like to print and then click Open. One or more objects should now be placed on the build plate.
4. Change the position of the object on the build plate by clicking and dragging it around.
5. Change the orientation of the object (if necessary).
    a. Clicking the Orient icon .
    b. Click on the +90 and -90 buttons or enter custom angles.
    c. A specific face can also be defined as the reference for orientation by clicking the Place Face on Build Plates button.
    d. The part should be oriented such that it uses the least amount of support material possible and any critical faces/features do not have support on them.
6. Check the scale of the part by clicking the Scale icon . Ensure it's the desired size. To change the size, click on the scale % boxes or the mm boxes.





7. Change the print settings.
    a. Click on the gear icon to bring up the print settings toolbar.
    b. Click on Custom Settings. Click on Quick Settings. The important settings to take note of here are the layer height, infill density, and the number of shells. They can be changed if necessary.
    c. If the part needs support, then click on the Supports + Bridging tab. Scroll to the bottom. Under Support Type, select Breakaway Support from the drop-down menu.
    d. At the bottom-right of the window, click Done. Click on the gear icon to hide the settings.



8. Click on the clock icon to start estimating a print preview. The software will now slice the object(s) into layers. Once it's finished a slider will appear on the left of the screen that can be moved up and down to show how each layer of the object(s) is printed.
9. Save the project. Click File, then click Save As… and choose a name and location.
10. Print the project. At the bottom right of the screen, click the Print button, then make sure the build plate is clear and click Start Print.

Sending your file by USB

11. If sending your project through a USB stick, the file needs to be saved with all required print settings, as a **.makerbot** file.
12. Insert the USB stick in the designated area on the printer, select the file you want to print and hit start. Once the print has started, the USB stick can be removed as needed.

Placing Multiple Prints on The Build Plate

Opening several files may cause some of the objects to be placed on new build plates. They can be copied to the first build plate by:

1. Clicking on the build plate they are on
2. Clicking on the object
3. Pressing Ctrl+c
4. Clicking on the first build plate
5. Pressing Ctrl+v

Common Issues

Lifting

Lifting is an extremely common problem on both Makerbots. They don't have heated build plates so the tendency of prints to lift depends more on environmental factors such as the room temperature and humidity. Winter conditions provide a higher susceptibility to lifting due to cooler indoor temperatures. Some quick fixes are to apply new painter's tape to the build plate (Replicator 5th Gen) or to clean the build plate with isopropyl alcohol (Replicator+). On the Replicator+, if even small prints are lifting, it may be time to replace the adhesion sticker on the build plate. If these fixes don't help with build plate adhesion then it's best to print the job on an Ultimaker.

Under Extrusion

There's a fairly common problem with the Replicator+ where, when printing the first layer of filament, it will throw a "filament slip" error, recover from it automatically, then repeat the error several more times. Sometimes it will go through enough cycles of the error that it will cancel the print. The "filament slip" is due to a bend in the build plate that causes the nozzle to be really close to the build plate in a certain location, preventing filament from being extruded. Since the Replicator+ can detect if filament is moving through the nozzle, it throws an error. The quick fix for this is to remove the build plate, clear any small bits of plastic from under it, replace the build plate, and restart the print. The long-term fix is to replace the build plate.

No Extrusion

The Makerbots can sense when there is no more filament left and will conveniently unload the remaining bit of filament and pause the print. More filament can then be loaded and the print can be resumed.