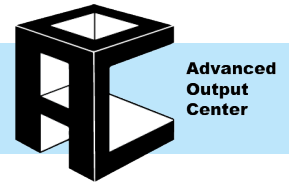


Guide for **Ultimaker 3 / S5**

Self-service 3D printing at the AOC



Advanced
Output
Center

General Information

Location:

Advanced Output Center, 1232 Sullivan (see page 9 for full list of Ultimakers on campus)

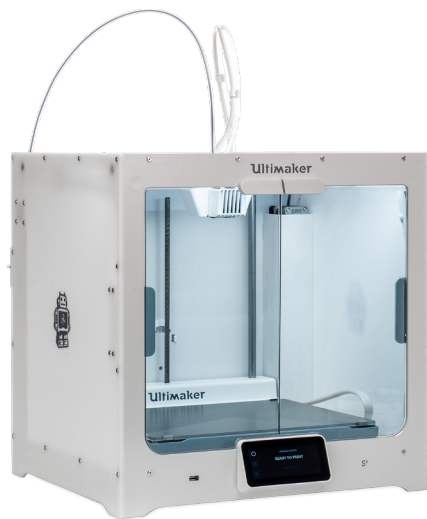
Access:

Certification required. You can walk-in or reserve the Ultimakers by talking to AOC staff. You must pre-slice your file before talking to staff, and show it to staff, so all check-outs and reservations must be done in person.

The certification course can be completed by searching or it on campus or through the link on our website.

Potential Fines:

All checkouts are given an extra hour for setup/cleanup. If you do not return on time there is a \$20 late fee. Please make sure you are returning equipment on time so the next patron can use it



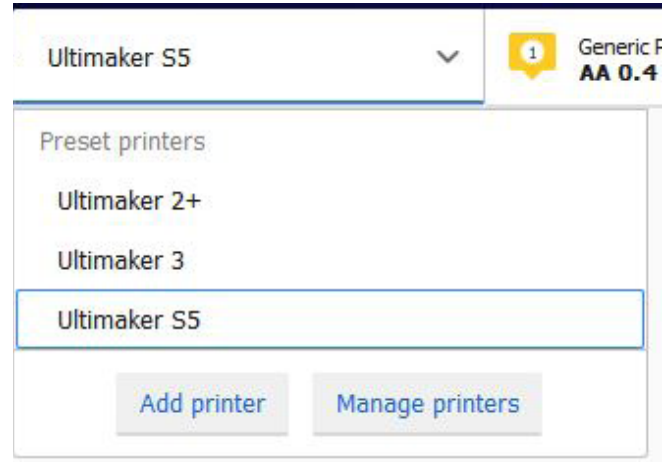
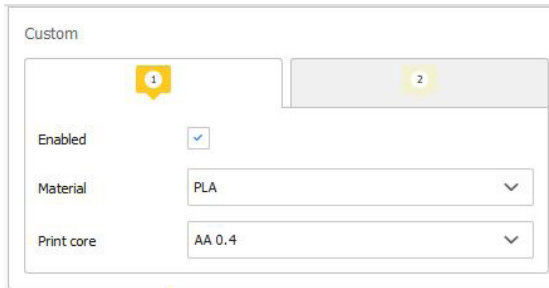
	Ultimaker S5	Ultimaker 3
Max Build Size	13"x9.4"x11.8"	8.5"x8.5"x7.9"
Resolution	200-20 Micron *using 0.4 nozzle	200-20 Micron *using 0.4 nozzle
Material	2.85 mm PLA only*	2.85 mm PLA only*
Leveling	Automatic Active Leveling Manual Leveling if Required	Automatic Active Leveling Manual Leveling if Required
Pricing	Free to use: Must provide your own PLA	Free to use: Must provide your own PLA
Software	Cura	Cura
File Types	STL recommended OBJ, X3D and 3MF possible	STL recommended OBJ, X3D and 3MF possible

Using Cura

To prepare your model for printing with the Ultimakers you will need to use Cura to “slice” your file. The term “slice” refers to using Cura, or other 3d printing software, to create a layer-by-layer tool-path file that the printer will use to create your object. To begin that process for the Ultimaker, open Cura and do File --> Open File(s)...

SET WHICH ULTIMAKER YOU ARE USING

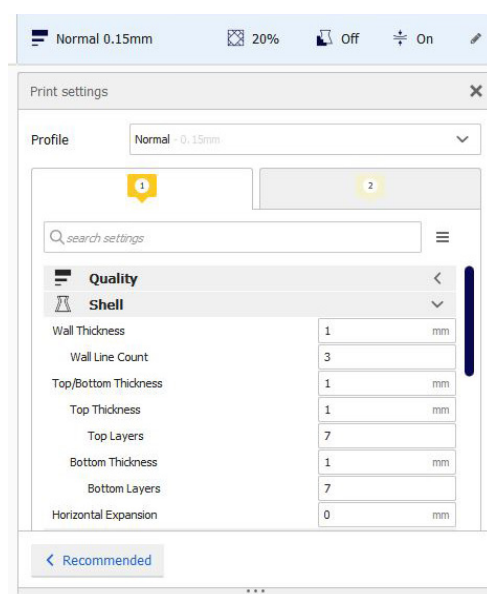
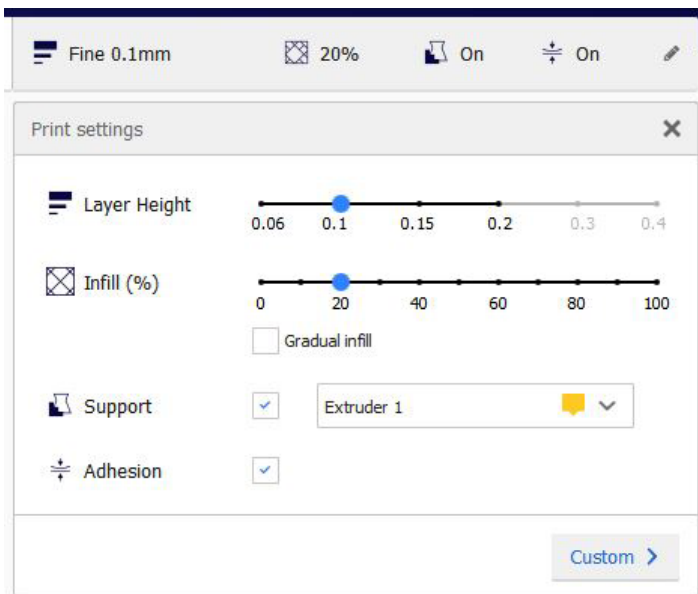
In the upper left of Cura you will see Ultimaker 2+, 3 or S5 on a white drop down menu. Before doing any more work on your file, make sure you are set to whichever Ultimaker you are going to be using. In the next panel to the right, you will be able to set what material and print core you will be using. For all prints in our lab, this should be set to Generic PLA for material and AA 0.4 print core. It is best to just use extruder 1 for this.



PRINT SETUP

The next option to the right is Print Setup. Here you can determine if you will use a preset profile or create a custom profile for your printer settings. For preset profiles you can choose between Fast, Normal, Fine or Extra Fine Resolution. To do this drop down the menu, select “custom” at the bottom, and then select from the presets in the drop down menu. We do not recommend changing any other settings in the custom view, but in the recommended view you can move the sliders to adjust the print settings to your preference.

The Fast preset will be the shortest print time but will give you the lowest quality result, where as Extra Fine will give you the highest quality result but will take longer to complete. If you would like to see the differences, the AOC has a sample board with examples of each resolution, located just to the left and above the Ultimaker shelving.



Using Cura

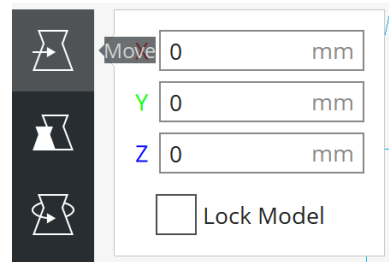
ADJUSTING THE MODEL

Once your model is loaded onto the bed and you have set your Print Setup settings, you now have an opportunity to adjust the model using the tools built into Cura. To use any of these tools make sure that your model is selected and then click the tool from the row on the left.

MOVE

Move your model by:

- Clicking your model and dragging.
- Entering exact X, Y, or Z coordinates.

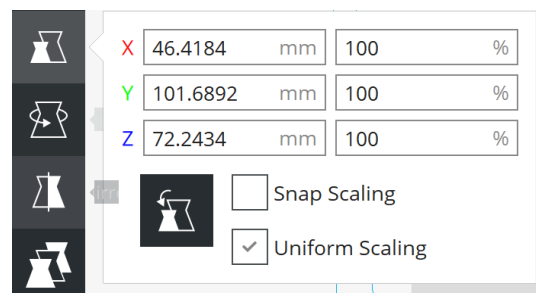


SCALE

Scale your model by:

- Dragging the X, Y, or Z handles.
- Setting a specific X, Y, or Z dimension.
- Setting a specific X, Y, or Z scale %.

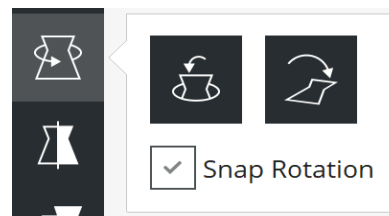
To scale uniformly, enable **Uniform Scaling**
Reset your model to its original size using **Reset Scale**



ROTATE

Rotate your model by dragging one of the rings along the axis.

Disable snap rotation for precise rotation by holding the shift key while dragging.



MIRROR

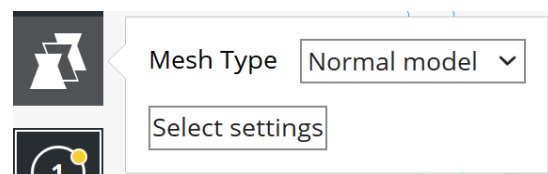
Mirror your model by selecting one of the six arrowheads in the desired direction.



PER MODEL SETTINGS

If you are printing multiple files on the same build platform you can define specific settings for each file using the

Per Model Setting feature. Select the desired model, select the **Per Model Setting** icon, then click **Select Settings**. These settings will only affect the selected model.



Using Cura

PRINT SETTINGS OVERVIEW

Below you will find descriptions for various settings you can change in the print settings tab. We recommend only using the the preset profiles in Cura and adjusting the sliders in the recommended view. Changing settings in the custom view can lead to print settings more likely to fail.

QUALITY

The Layer Height, also referred to as resolution, is the thickness of one printed layer (in mm). With a thinner layer height you will increase the quality of the print. Using thicker layers can decrease the print time substantially. Cura offers layer heights between 0.06 mm and 0.3mm.

SPEED

Speed determines how fast the print head will travel during printing and should be correlated with the nozzle heat. A print speed about 75 mm/s should not be used because the print is likely to fail and can potentially damage the printer. To make the print profile faster you can use less shells, thicker layers, less infill, or brim instead of raft (using a brim instead of a raft does have a higher risk of failure, at least on AOC printers due to lab environment conditions.)

Recommended Speeds:

40-50 mm/s **Print Speed** for most prints.

Nothing over 75 mm/s Print Speed and 150 mm/s Travel Speed.

SHELL

Wall Thickness:

The nozzles on our Ultimakers print lines that are 0.4 mm wide. In general a wall thickness of 2 or 3 times the line width is sufficient. A higher value will create a sturdier model while a lower value can significantly decrease the print time and filament usage.

Recommended Wall Thickness:

0.8 mm for most prints. 1.2 - 1.6 mm for sturdier prints.

Top/Bottom Thickness:

Top/Bottom thickness allows you to set the thickness of the solid printed top and bottom layers of the print. A higher value ensures all gaps on the top and bottom layers get closed completely. Always use a multiple of your layer height when setting your top/bottom thickness.

Recommended Top/Bottom Thickness:

At least double your layer height.

INFILL

The Infill density defines the amount of filament used on the inside of the print. A higher infill density means that there is more plastic on the inside of your prints, leading to a heavier and stronger object.

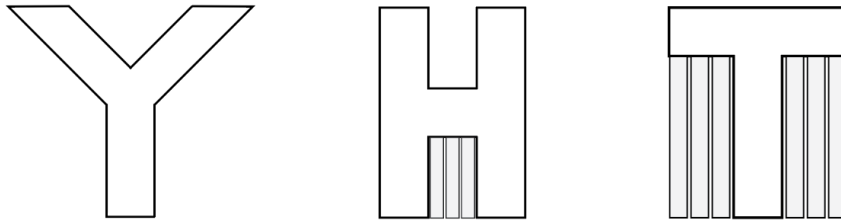
Recommended Infill:

10% infill for models intended for a visual purpose. 20% or more for models with a structural purpose.

Using Cura

SUPPORT

Some models have overhanging parts, meaning parts of the model would float in mid-air when the machine is printing. Unsupported parts like this will almost always fail as the filament has nothing to bond to. Enabling support structure under the model can prevent the filament from printing onto nothing and falling. It solves this problem by building sacrificial support structures under overhangs. Typically these structures are made of the same filament that is being used to create the main print. Supports can be placed “touching the build plate” or “everywhere”. Support is recommended on any prints that have overhang angles that are greater than 45 degrees.



Use “touching the build plate” if your object has simple overhangs with nothing below them.

Use “supports everywhere” if your model has more complex features, such as an overhang being located over a previously printed part of the model.

BUILD PLATE ADHESION

Skirt:

A skirt is a line printed around the object on the first layer, but not connected to the object. This helps extrusion and can also be a good check for bed leveling before the print starts.

Brim:

Brim adds a single layer flat area around the base of the model, like the brim of a hat. Brims can prevent warping of your model by creating a larger bottom surface area to help your model adhere to the build platform.

Raft:

A raft adds a thick grid between your model and the build plate. For models that have multiple small points that contact the build plate, rafts allow for easier detachment of your model, but do increase your print time and material usage.

Recommended Setting:

We always recommend using some form of build plate adhesion when printing at the AOC. For most prints, a brim will be enough adhesion and use the least material, making it the preferred option. For larger prints, we recommend you always print with a raft and set the **Raft Air Gap** setting to 0.4. These settings will give you the most stable results and make it easy to remove your model at the end. Using skirt or no build plate adhesion increases the potential risk of failure.

SAVING YOUR FILE/CHECKING OUT THE MACHINE

Once you have adjusted and are happy with all of your settings, you will need to save your file. You will need to show an AOC worker your file so we can verify it uses the correct settings and check the print time. You will then receive the key and USB drive when you are ready to print. Once you plug the storage device into the computer you will see a **Save to Removable Drive** button in the bottom right. You can either do this or **Save to File** to put the print file on our storage device. We are now ready to prepare the printer for printing.

Preparing The Printer

BUILD PLATE LEVELING

Leveling is the process of setting the distance between the extruder nozzle and build plate which can significantly increase print quality and prevent damage to the machine. Leveling is performed two different ways on the Ultimaker 3 and S5, either through Active Leveling or Manual Leveling. We will cover both here, but Active Leveling should be your first option and you should only attempt manual leveling if you are encountering issues after active leveling.

Active Leveling:

To use **Active Leveling** on the Ultimaker 3, go to **System - Build-plate - Active Leveling**. On the Ultimaker S5 you can follow the same steps by following the icons for **Machine - Settings - Leveling**. Once engaged, Active Leveling will have the head move to positions around the build-plate, adjust the height accordingly and leave the build-plate in a well leveled position.

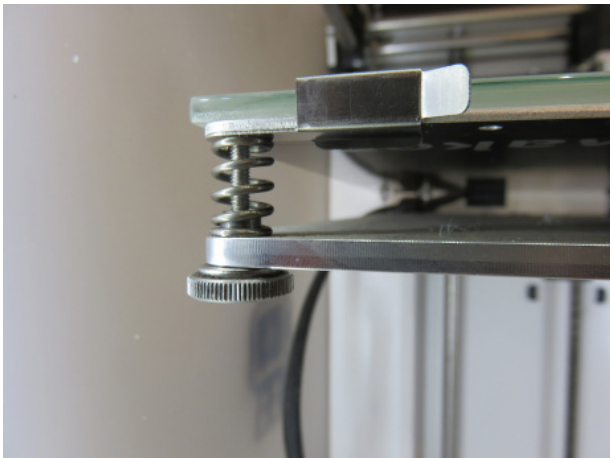
Note: The Ultimaker printers in the AOC are set to auto-level before every print, so you do not need to engage active leveling when you start a print.

Manual Leveling:

If you are still seeing issues with printing after automatic leveling, you may need to try **Manual Leveling**.

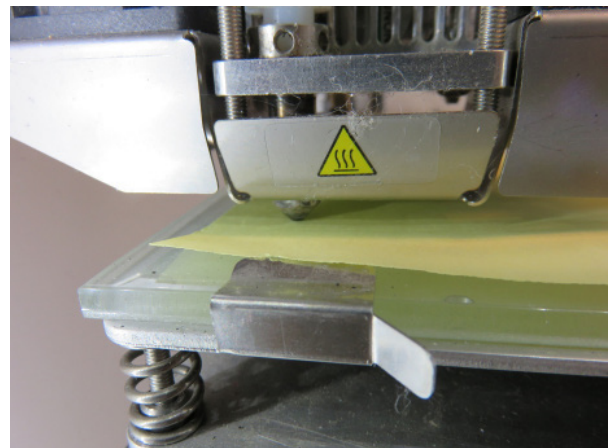
To begin leveling, select **Maintenance** than **Build Plate** on the printer then follow the prompts carefully. If you have any issues or questions during this process please ask an AOC worker for help if this is during open hours.

The machine will prompt you to raise the build platform until it is a millimeter away from the nozzle. This measurement doesn't need to be exact, however, the extruder should not be touching the build plate and the extruder should not be too far from the build plate either. To fine tune, we recommend to use a post it or similarly thin piece of paper between the nozzle and the build plate. For this step you should feel friction on the paper as you slide it. If there is too much resistance and you cannot slide the paper or you feel no friction at all you will need to adjust the screw directly under the extruder on the underside of the build plate. Turn **Clockwise** to move the bed closer to the extruder and turn **Counterclockwise** to move the bed further.



Use the screws on both sides of the build plate to fine tune the adjustment and match the desired level. Tighten the screw to lower the plate and loosen the screw to raise it closer to the extruder.

Use a normal piece of paper, post-it or similarly thin piece of paper to adjust the tension until you have the right amount of resistance from the nozzle on the paper when you move it.



Preparing The Printer (Continued)

LOADING/UNLOADING FILAMENT

Once the bed is leveled we can move on to preparing our filament for printing. The self service printers at SAIC all use 2.85 mm (you will often see this rounded up to 3 mm) PLA, a biodegradable plastic that is manufactured using plant-based materials like corn starch or sugar cane. You can purchase rolls of filament through SAIC resale, including the one in the Sullivan Center Wood-shop (rm 1240*), or you can source it yourself from various vendors.

Filament Recycling:

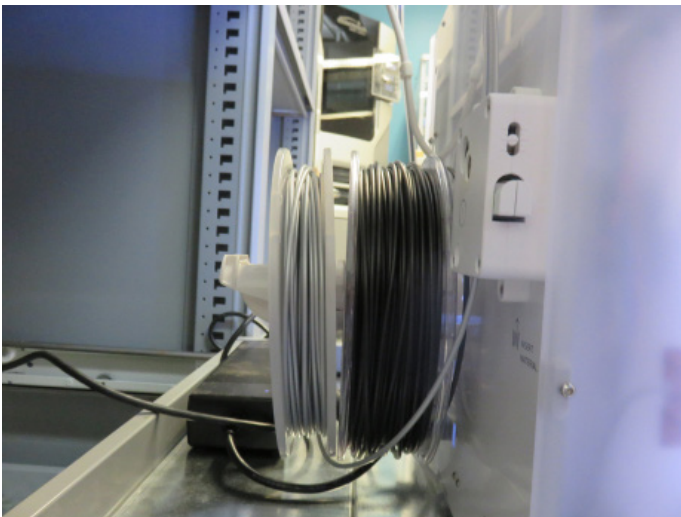
Because PLA is biodegradable, it is also recyclable. In every SAIC lab with Ultimakers there should be a gray bin marked for PLA collection. All PLA waste from your print can be deposited here, including filament scraps, rafts and brims, support structures and even failed prints.

Filament on Ultimaker 2+:

Loading and unloading filament on the Ultimaker 2+ is done with a single command. To start go to **Material** then **Change**. Once started the nozzle will begin heating up and then it will reverse any current filament out of the machine. If there is no filament loaded simply wait until it prompts you to enter your material into the back. Make sure you have a clean edge, cut off any melted or jagged ends to your filament, and then insert the filament into the back until you feel it catch on the wheel and slight movement from the motor. Once you feel that movement you can confirm on the menu. The printer will now push the filament from the feeder, up the bowden tube and to the nozzle of the printer. Wait until your filament is being extruded from the nozzle and then press confirm.

Filament on Ultimaker 3:

The 3 has three different ways to interact with filament. **Change** will do the whole process outlined above with the 2+, where as you can also just **Unload** or just **Load** instead of doing them both. To access these options go to **Material/Print-core** and then **Material 1** (or **Material 2** if you are using extruder 2 or both extruders). Once you are in these options the process works the same way as the 2+ so refer to the 2+ filament instructions above and follow the prompts on the screen. For **Material 1** you will insert your material into the feeder on the left of the machine, while **Material 2** will use the feeder on the right.



Ultimaker 3 loading

In this photo the roll on the right (black) is **Material 2** and the roll on the left (silver) is **Material 1**. The feeder for **Material 2** is on the far side of the filament in this photo, on the right of the machine if you are facing it from the front.

Printing on Ultimakers

STARTING YOUR PRINT

To start your print make sure the USB drive, with your file on it, is inserted into the front of the machine. Select Print and then select your file. Be sure to stay with your print for at least the first 10-20 minutes of the print. Most failures that occur on these printers happen at the beginning of the print so staying around for the beginning of the print is vital.

The estimated print time on the Ultimaker is always incorrect as it estimates the build based on the current movements it is making, not the total information of your build. Because of this, ignore the estimate on the machine and use the estimate that you got in Cura before you started printing. This time is much more accurate and your checkout time at the AOC will be determined by how long Cura estimated, not how long the Ultimaker estimates.

REMOVING YOUR PRINT

Once your print is completed you can simply remove it from the build-plate. It should be easy to remove the print as long as you used our AOC preset profiles while printing. Do not remove the glass build plate, if you are having issues removing your print yourself please ask AOC staff for assistance.

RETURNING YOUR CHECKOUT

When you are finished you need to remove your filament, lock up the printer and return the key and any storage devices checked out back to the AOC. If you have more to print or need to extend the time talk to an AOC staff member about extending or making a new checkout for the device. We only have four Ultimakers at the AOC so at high traffic times during the semester you may not be able to renew or check out the Ultimakers again if there are patrons waiting to use them.

Printing on Ultimakers

3D PRINTERS AROUND CAMPUS

If you cannot use the Ultimakers at the AOC there are also the following options around campus for Ultimaker use.

Open Access:

Columbus Digital Fabrication Studio | Columbus, Main Floor

General Access Lab | MacLean, 9th Floor

Department Only Access:

Sharp Instructional Shop, Digital Lab | Sharp, 3rd Floor
Contemporary Practices only

IO Lab | MacLean, 4th Floor
Art and Technology Only

USEFUL SOFTWARE, EQUIPMENT AND WEBSITES

Tinkercad:

Easy to use 3d CAD design tool. Based on-line, there is no software to download and it is very easy to learn.
tinkercad.com

Meshmixer:

Software for cleaning up 3d scans, making a water tight model for printing, and digital sculpting.
meshmixer.com

Netfabb:

Easy model repair for cleaning up and repairing 3d files for printing.
autodesk.com/products/netfabb

Thingiverse:

Community of 3d file sharing, make sure to save files as .STL
thingiverse.com