STANFORD
PRODUCT
REALIZATION
LAB

Ultimaker II Tutorial

You MUST complete this tutorial before using the Ultimaker II for the first time.

Overview

The Ultimaker 2 (UM2) is an additive manufacturing (AM) machine – more commonly referred to as a 3D printer. The UM2 uses Fused Filament Fabrication (FFF) technology to create physical parts directly from 3D computer-aided design (CAD) models. This is the same technology that is utilized by the Stratasys Dimension FDM machine also located in Room 36.

Design Guidelines and Considerations

Material

The UM2 uses a mix of PLA/PHA, both biopolymers derived from plants. This thermoplastic is relatively rigid yet tough, is biodegradable, and comes in a variety of colors. In general, PLA is stiffer than the ABS used by the Dimension, but less brittle than the acrylate photopolymer used by the ProJet.

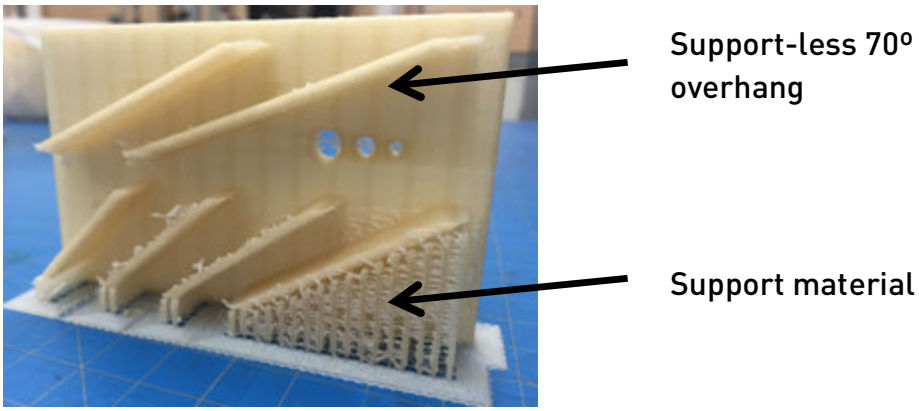
Resolution

The resolution of an FFF printer has two main components: layer height (Z-direction) and feature resolution (XY-direction). The layer height is simply that: the thickness of each layer of plastic deposited.

The UM2 has an adjustable layer resolution between 0.06mm and 0.25 mm. The feature resolution is determined by the nozzle diameter of the machine (the machine deposits a path of plastic narrower than the nozzle). Since the UM2 has a 0.4mm diameter nozzle, the smallest features in can reasonably reproduce are around twice that: 0.8mm.

Support Material

The FFF process necessitates the use of support material because of the layered approach to construction. Flat-bottomed cantilevered sections can be printed without support material, but there will be poor surface quality. In general, overhangs greater than 60 degrees off vertical should have support – although in the photo below, the top right cantilever at 70 degrees off vertical printed fairly well without support.



The UM2 has a single nozzle and a single material. Therefore, the system uses Same Material Support Material (SMSM), otherwise known as Breakaway Support Material (BSM). The machine perforates the interface between the support and model, making the support easy to tear off afterwards.

Finishing

The first step in finishing is to remove any excess material generated in the printing process (support material, brim, raft) with small pliers and side-cutting snips. The parts can then be sanded or shaved down to remove further traces of the support material and to reduce the appearance of layer lines. Parts can also be painted via standard methods (primer recommended).

Creating a 3D File

3D printers require a digital model to print. These models can be generated from scratch in a CAD program like SolidWorks, they can be generated from 3D scans, or they can be downloaded and modified from the internet. No matter what type of file you begin with, you will need to end with an .STL file (**S**tandard **T**esse**L**lation **F**ormat).

From CAD

You can use any CAD software that creates 3D geometry, including SolidWorks, Rhino, Autodesk 123D and Inventor, SketchUp and more, though you will need to be careful to export as an .STL file. *(Note: Some software, like SketchUp, will not export to .STL, so instead you can export as .OBJ and then use another program like NetFabb to convert to .STL.)* You can export either single parts or assemblies of parts. Depending on the software, you simply need to go to *File*, then either *Save As* or *Export* and make sure to select **.STL** as the format. CAD programs will provide some options for the export, so here are some recommended settings:

File Encoding:	Binary (not ASCII)
Resolution/Deviation:	0.4mm
Units:	Millimeters

From Scans

We have two scanning technologies in Room 36. The DAVID SLS-2 is a high-precision structured light projector and camera scanning setup located in the corner. Autodesk 123D Catch is a free mobile application that uses photogrammetry to create scans. Both will allow you to export/download an .STL file of the scan. Often, scans will have holes or other discontinuities that will affect printing. Programs like Netfabb and MeshMixer can aid in the repair of these errors.

From the Web

There are an increasing number of sources on the web for digital models. You can find raw CAD models on sites like GrabCAD. You may also find .STL files, many of which were

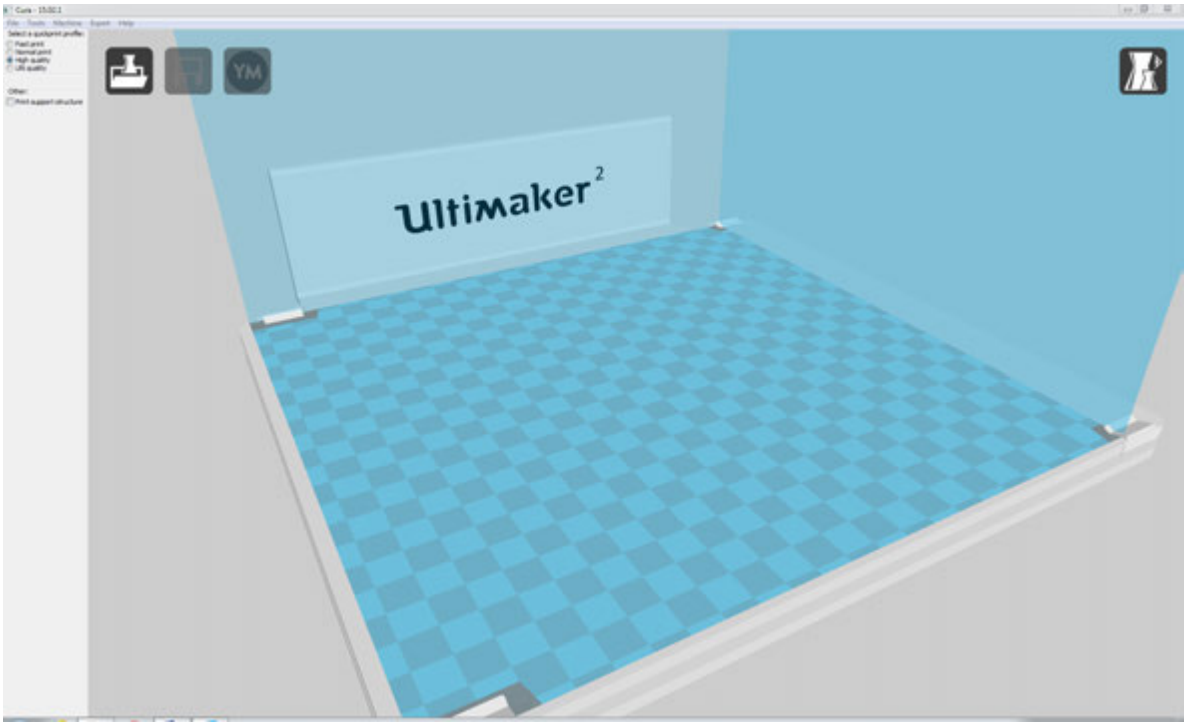
designed specifically for additive manufacturing, on sites like Thingiverse and Youmagine. These models can be excellent jumping off points for your own designs. You can make geometric modifications by importing the .STL into SolidWorks or you can make organic modifications by importing the .STL into MeshMixer, a digital sculpting tool. **REMEMBER:** *you should always cite the source for any part you have used or modified!*

Preparing the File for Printing

The next step requires a program to convert the 3D geometry of an .STL file into code that will tell the UM2 how to operate. This process is called slicing and the resulting file is called g-code. We use a program called Cura, which is free and open-source.

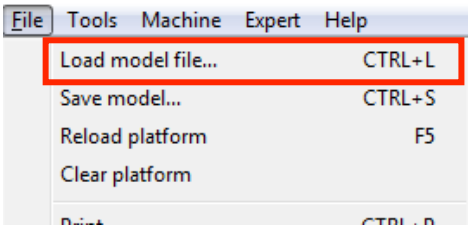
Open Cura

A shortcut to Cura can be found on the Desktop of the UM2 computer. Double-click on the icon, shown to the right, to start Cura. (If there is already a Cura window open, please close it and begin anew). The Cura window will then pop up and show an empty printer bed. Below is a screenshot of that window:

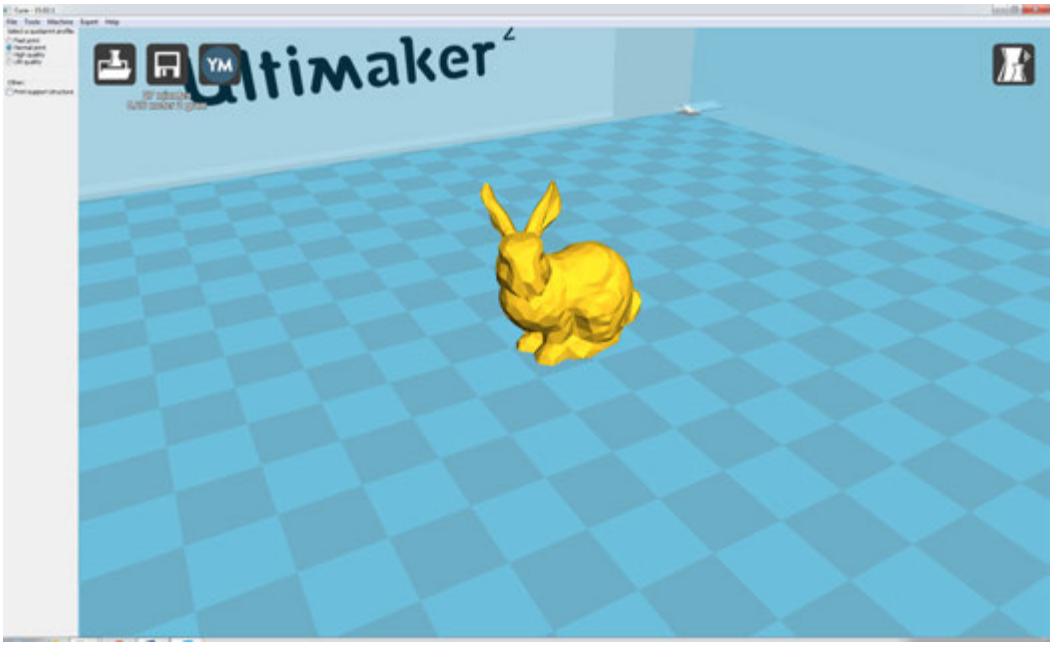


Load Your File

To load your STL file, either go to *File, Load model file...* or click on the Load button. If you are doing the tutorial, you should open the **UM2_TUTORIAL_PRINT.STL** file that is located on the Desktop.

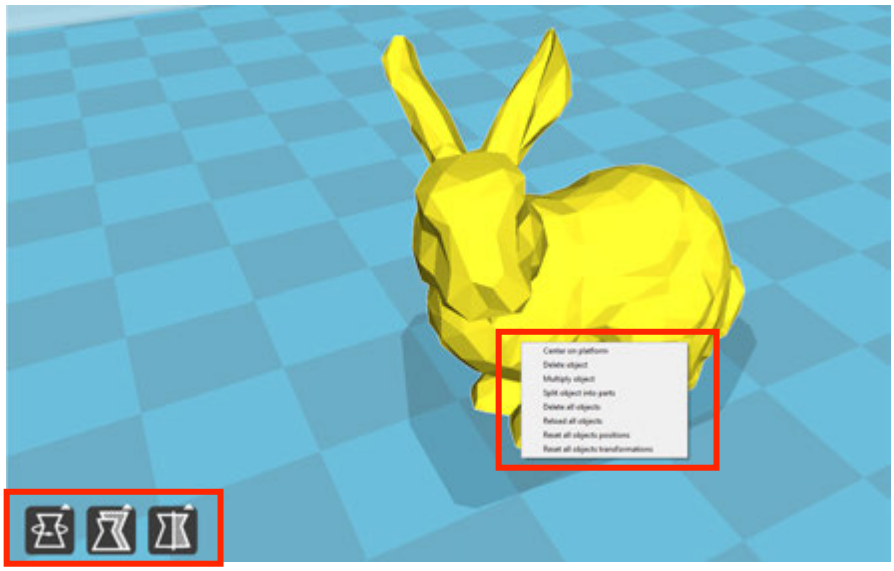


Once you open the file, it should show up on the build platform. Perform a sanity check to ensure that your model is the proper size. Mismatched export and import units could cause incorrect sizing. You can fix this by using the Scaling function (see below). You may also load multiple models onto the build platform at any time.



Modifying Your Model

Once you click on a model, three tool options appear at the bottom left corner of the screen: Rotate, Scale, and Mirror. You can use these to make adjustments to your model. Right clicking on the model will also provide you some options like creating copies of a part, splitting an assembly of parts, and more.

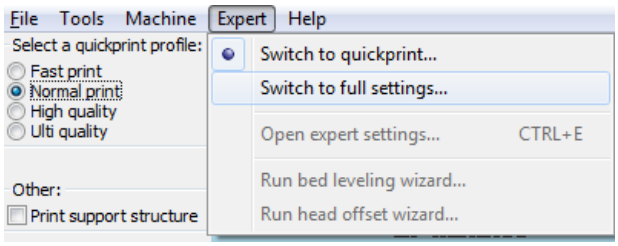


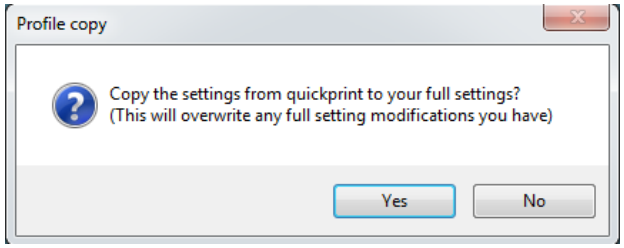
Choosing Your Print Settings

The Cura software provides the user with many more options than our other 3D printers. This freedom and flexibility can be great, but it also puts more responsibility on the user to understand the implications of their choices. Since everyone has different levels of experience in additive manufacturing, we have both standard settings that should accommodate 90% of use cases, as well as custom settings for advanced users. Advanced users should note that this machine is fiddly and small deviations affecting the extrusion rate could result in failed parts and downtime for the machine.

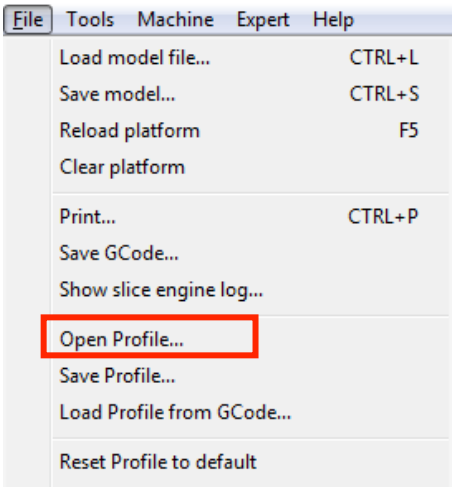
For example, in the **Tools** menu, DO NOT be tempted to choose **"One at a Time"**. Your second part (and those that follow) will not adhere to the buildplate and will fail.

Start by switching from the Quick Print mode to the Full Settings mode by going to *Expert* and clicking *Switch to full settings…* When it asks you whether or not to copy settings over, choose **No**.





Once you are in the Full Settings mode, you can load the default settings profile by going to *File, Open Profile…* You should navigate to the Desktop and select the **UM2_DEFAULT_PROFILE.ini**.

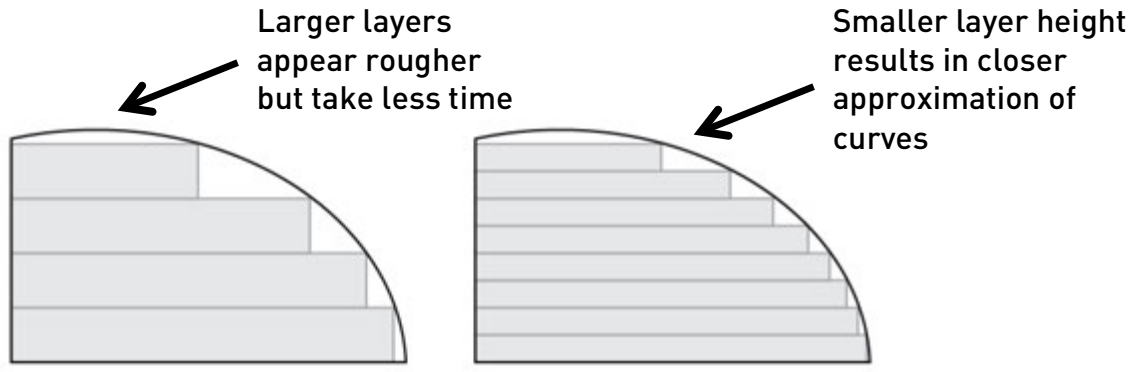


There are only four settings that you will manipulate:

- **Layer Height (mm)**
- **Fill Density (%)**
- **Support Type**
- **Platform Adhesion Type**

Layer Height

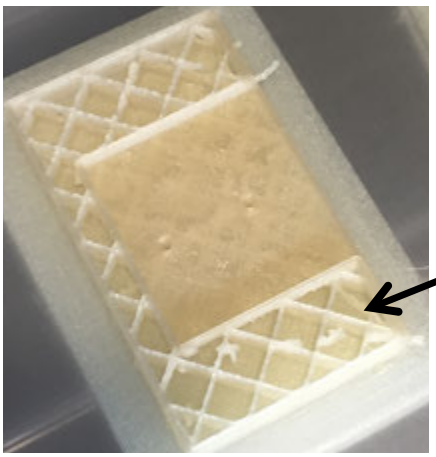
Determines the resolution of the model in the z-direction. A smaller layer height means a higher resolution, higher quality model. However, the smaller the layer height, the longer the print will take, so you must make a tradeoff.



Recommended Range: **0.08 mm - 0.15 mm**

For Tutorial: **0.10 mm**

Fill Density



Part interior cavity is filled with a lattice structure of specified density

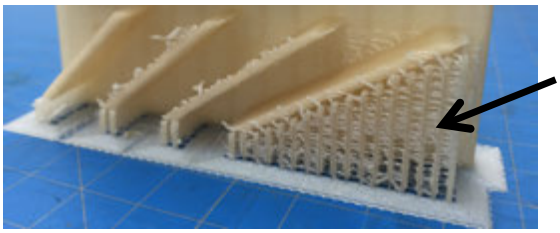
FFF technology can build the model with a non-solid interior. The software generates a lattice structure to fill the inside of the part and you get to choose the density of the lattice. The sparser the lattice, the lighter, cheaper, faster, and weaker the part is. The denser the lattice, the heavier, more expensive, slower, and stronger the part is.

Recommended Range: 10% – 30%

For Tutorial: **15%**

Support Type

Cura provides three options for generating support structures under overhangs:



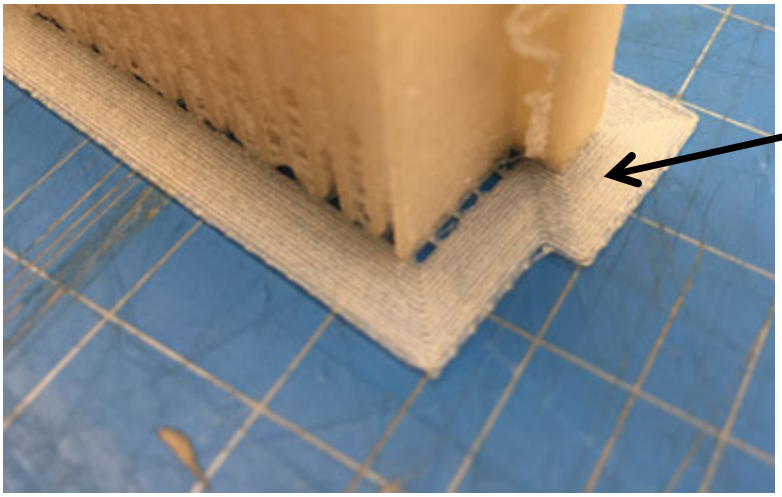
"Touching buildplate" support only generated for overhangs greater than 60° projected directly over the buildplate

- ***Touching Buildplate****:* Recommended. Overhang areas greater than 60° that project down onto the buildplate will receive support. Overhang areas that project down onto other parts of the model will not receive support.
- ***Everywhere:*** Safe. All areas with overhangs greater than 60° will receive support. A conservative option but will make cavities like through-holes hard to clean.
- ***No support:*** Not Recommended No support material will be created. Only use if you are extremely confident that your model does not require it.

For Tutorial: **Touching Buildplate**

Platform Adhesion Type

You have the ability to choose how the part interfaces with the glass build platform:



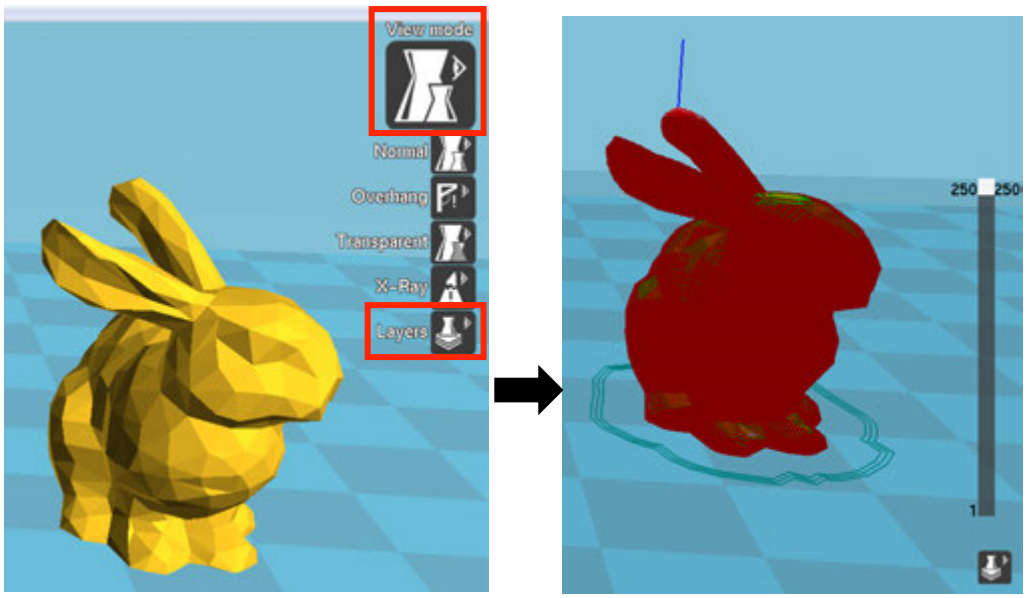
"Brim" adds a few extra laps around your part to increase surface area & improve adhesion

- ***Brim:*** Recommended. Creates a many-ringed, single-layer extension of the base of the part that can be torn/cut off afterward. Ensures parts will remain stuck to the bed during printing and prevents warping of sharp corners. The photo above shows a brim around the part.
- ***Raft:*** Safe. Creates a platform of support material underneath the entire 'shadow' of the part. Good for large parts with tiny footprints.
- ***None:*** Not Recommended. Only good for large base areas and no sharp corners.

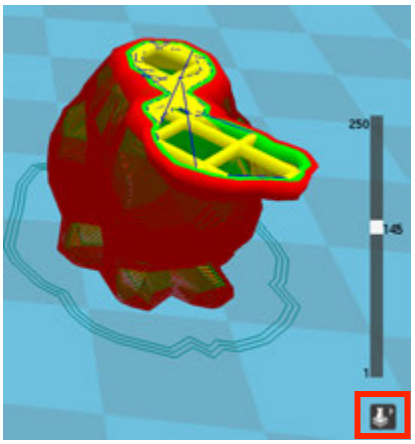
For Tutorial: **Brim**

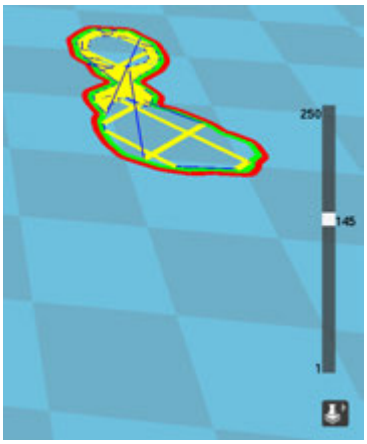
Viewing the Layers

Once you have selected a print mode, you can go to the Layers view option to make sure that your part will print as expected. After clicking on the *View mode* icon, four options appear. Click on the bottom-most icon *Layers* to enter layer view. Cura will take a few seconds to a minute to load the visualization. You will see the model in multiple colors and a scroll bar on the right hand side denoting the current layer # (on the right) and the total # of layers (on the left).

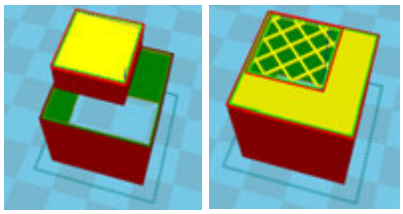


Once in the layer-viewing mode, you can drag the slider up and down to view different layers of the model. It will show you the current layer and everything below it. Clicking the small icon below the slider will switch it to single layer mode. Red is the outer perimeter of the part, green is the inner perimeter, yellow is infill, and teal is support.





NOTE: There is a rendering error that causes flat surfaces to appear hollow when viewing them from a higher layer. Don't fret!



If the model does not appear as expected, there is probably something wrong with your file.

Slicing the Model

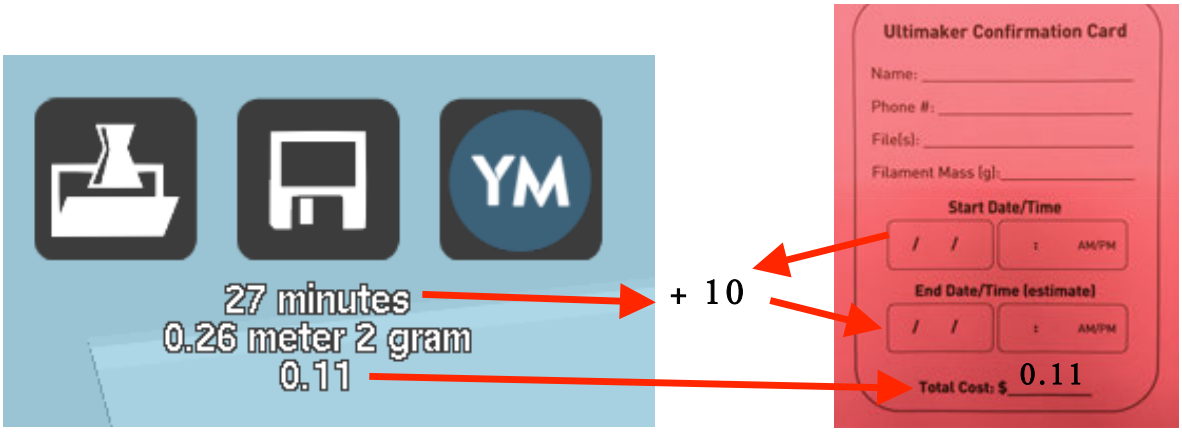
Cura automatically slices the model into layers and generates the machine's gcode whenever you make a change to the settings or to the model. Once you are happy with the layer view of the model, the print settings, the print time, and the amount of material, then you are ready to load the file onto the printer. First though, some housekeeping.

Logging the Print

Fill Out Print Confirmation Card

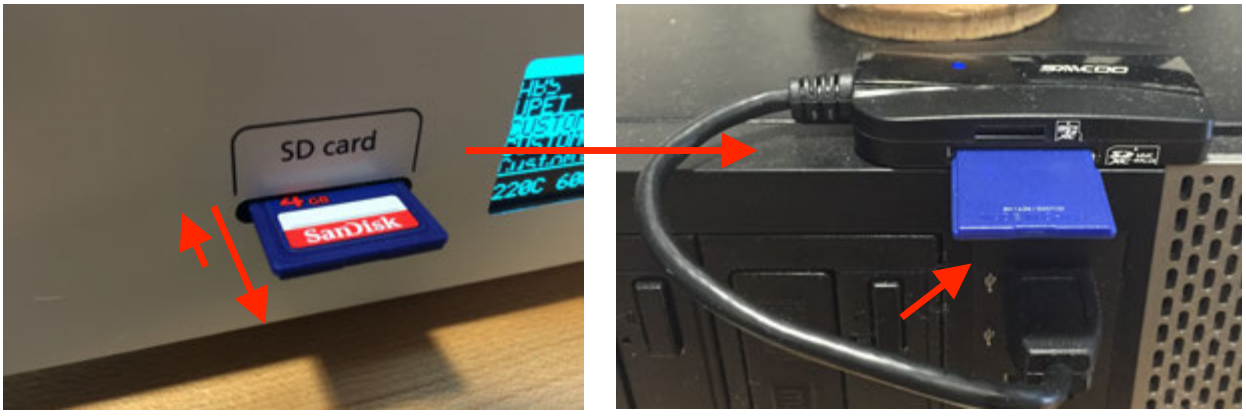
Print confirmation cards are located on the left side of the UM2. Please take a card and fill out the information requested on it. It is very important that you record the **Filament Mass** properly! We use this to determine whether your print will finish successfully with the current material spool.

In order to calculate the end time, please add the time estimate provided by Cura to the current time plus 10 minutes (for payment and warm up). This must not run into the next reservation period unless you have that reservation too! The Cura software provides the total cost on the bottom line of text. Yes, that is dollars and cents!



Saving the File to the SD Card

The Ultimaker2 uses an SD card to transfer files to and from the printer. Right now the SD card is probably in the printer from the last print job. Press in on the card and it will pop out. Place the card label-side down into the card reader plugged into the computer.

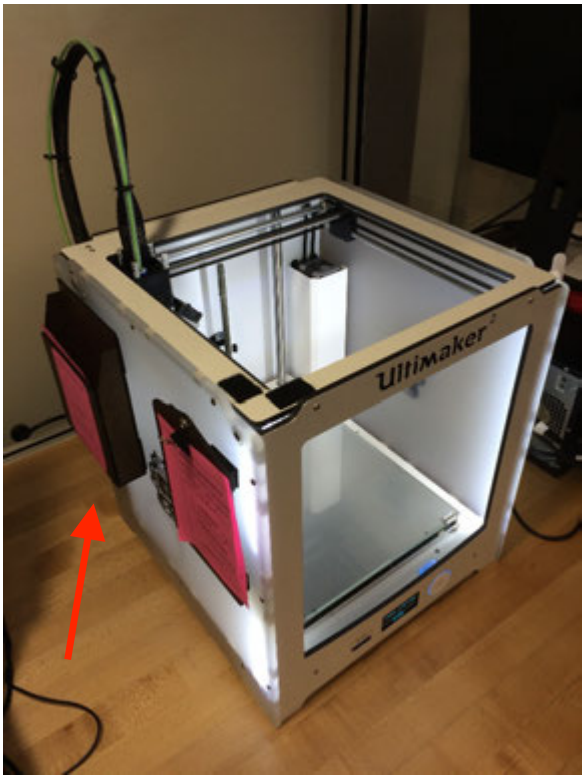


You can then click on the floppy disk icon in the upper left hand corner to save the gcode to the SD card. If there are other memory devices in use (like flash drives), Cura will ask you which device to save the file to. You can find out which letter drive is the SD card by opening a Windows File Explorer window and looking at the files present on the drive. (It's usually the G drive).

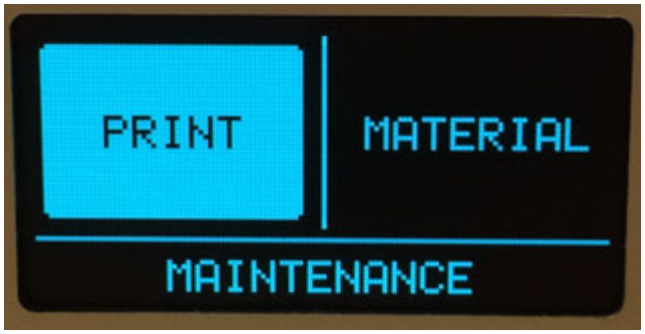
Preparing the Machine

Power

The UM2 has a small power switch located on the back left corner of the machine. If this is not already in the ON position, you should toggle it to the ON position.

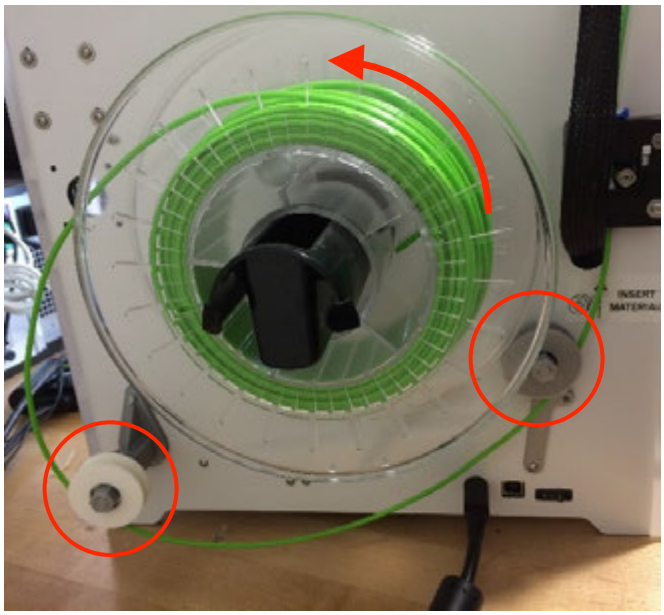


Once switched on, the UM2 will light up, the fan will turn on, and it will display a splash screen. When it is ready, the screen should look like this:



Material Check

Check to make sure that the material spool is loaded correctly and that the filament is being successfully guided around the two pulleys.



Print Bed

Please ensure that the bed is spotless before printing, otherwise you risk a failed print. If there are any small pieces of plastic stuck to the bed, use the razor-blade scraper with the red handle to remove them.

WEAR KEVLAR GLOVES.

If the bed looks oily or has residual residue on it, you can use an alcohol wipe to gently wipe down the bed.

AVOID TOUCHING THE BUILDPLATE WITH YOUR FINGERS!

First, the bed is heated and... well, it will be hot. Second, the oil from your fingers will prevent the plastic from properly adhering to the glass.

Paying for Your Part

Before you begin printing, you need to ensure that there will be enough material to finish the print job. To do this, we keep a print log. Bring your confirmation card to the TA and ask him/her to show you the print log in the TA cabinet. Do the math together to figure out if there should be enough material left for your print, and **do a sanity check at the machine.** If the spool looks threadbare, do not start your print with that spool. If necessary, have the TA unload the material and weigh the spool with the lab scale. The material mass remaining will be the total mass minus the mass of the empty spool (250g).

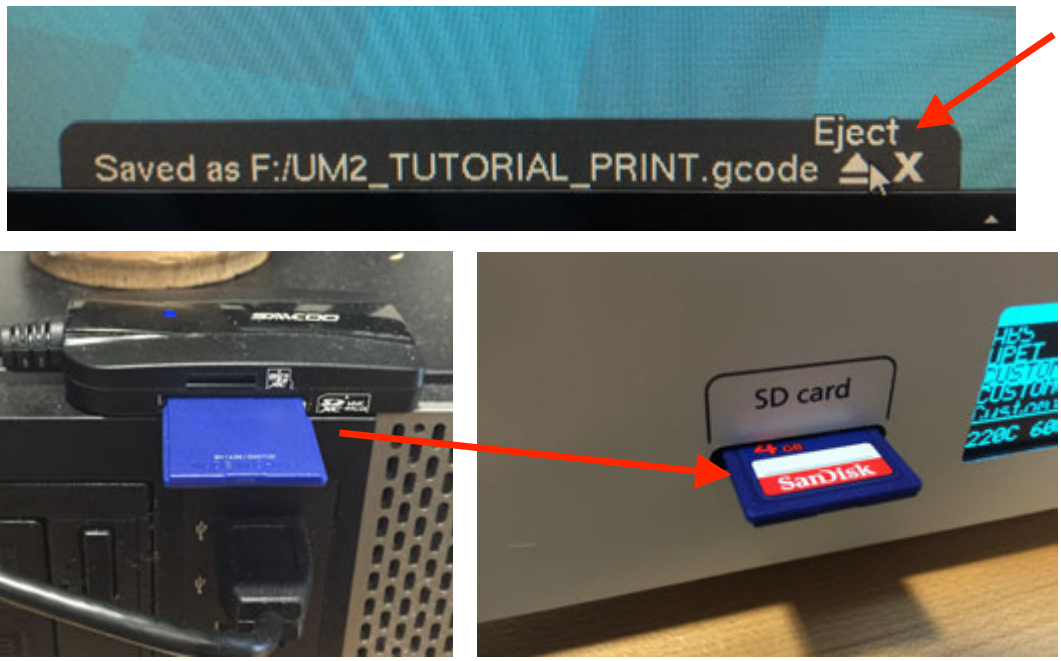
If your job's Filament Mass is more than the current spool material remaining, please ask the TA to replace the material spool.

The TA will then come over and check your file and machine setup as well as the end time estimate. If everything checks out, then you can pay for your part with cash or check. Once you have paid, clip your confirmation card to the side of the machine and start your print.

Printing Your Part

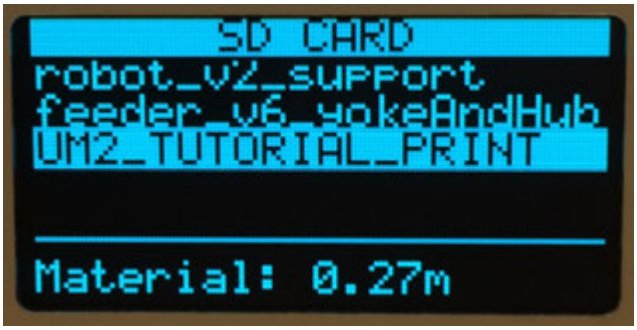
Insert the SD Card

Remove the SD card from the USB card reader, making sure first to eject the card via the software. Insert the SD card label-up into the machine's card reader, pressing in until the card clicks into place.

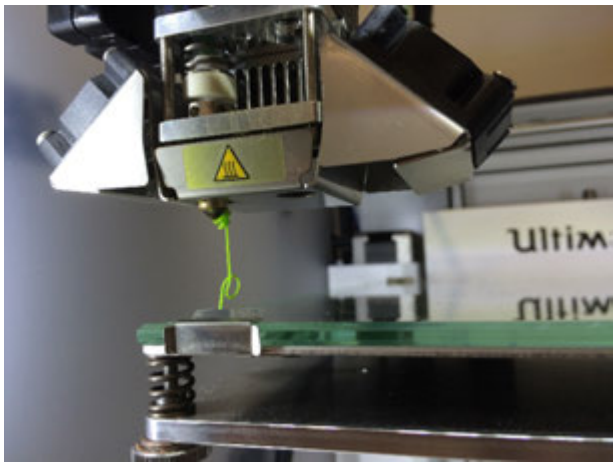


Start the Print

Using the scroll wheel, navigate to the **PRINT** option and click once. The screen should display all of the files on the SD card. The most recent file will usually be the last in the list, unless you updated a file of the same name that was already on the card.



The print head will move to the front left corner of the machine and the nozzle and bed will begin to heat up. Once they have reached printing temperatures, the bed will raise towards the top of the machine. The machine will then prime the nozzle by extruding a glob of material. Once this is finished, it will move to begin the print.



NOTE! The "Time left" value displayed on the screen is totally wonky. Never trust it.

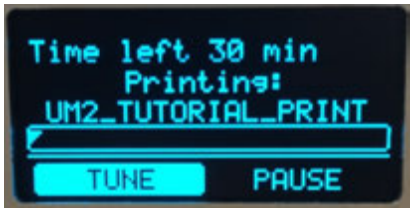
Watch the First Layer

Please watch most of the first layer of the print to ensure that there are no errors or issues. You should check for the following items:

- Was the pre-print priming extrusion successful (did plastic exit the nozzle)?
- Does it appear to be printing the base of your part and not someone else's?
- Is the plastic fully adhering to the bed, or is it pulling up and being dragged?
- Does the thickness of the first layer appear to be consistent across the base?

If the answer to any of these is NO, then please Abort the print.

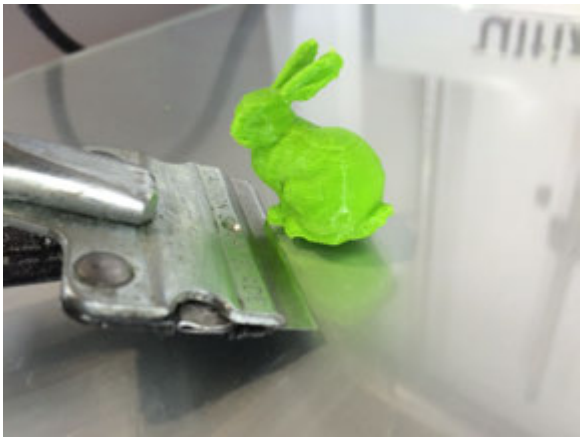
To abort the print, use the scroll wheel to navigate to the **TUNE** option on the menu. Then select the **Abort** option and click **YES**.



You should then take a picture of the part on the bed and make note of what the issue was. After the nozzle has retreated to the corner and the bed has fully lowered, you may scrape the plastic remains off the bed. Then please see the TA on duty for help.

Removing Your Part

Wait for the print bed to cool down for ≥ **2 minutes** before removing your part. Some parts may simply pop off the bed with a light force or twist applied to them. Parts with larger surface areas will require a tool to remove. Use the razor-blade scraper with the red handle to remove the part. **WEAR KEVLAR GLOVES**. Approach it from a few angles to get the best results. After removal, you can use tools like small pliers, flat-edge cutters, and deburring tools to remove the brim, support material, and to clean up the part.



Cleaning Up

Become a Registered User

If you have just completed the tutorial print, then go find the TA on duty and ask them to add you to the list of registered UM2 users.

Clean the Bed

Use the razor-blade scraper to make sure that the bed is free of any large plastic debris. If it looks greasy (or you just really feel compelled to!) use an alcohol wipe to remove any residue from the bed.

Clean the Build Chamber

Use the Swiffer duster and/or cleanup paintbrush to clean the area around the bed. From the home screen, you can navigate to **MAINTENANCE** and select **Raise Platform**. This will allow you to clean under the bed as well.

Clean the Workspace

Clean up any plastic debris around the workstation, put tools away, log out of computer accounts, and make sure to save your files to a flash drive, web service, or email.

Turn Off the Machine

Use the toggle rocker switch on the back of the machine to turn the UM2 off.