
CACM Printer Farm Setup

A student user guide for the Cura Slicer

Overview

This guide explains how students can prepare, slice and send their files using both Cura and 3D Printer OS

3D Printing Farm Policy

- 1) Allotments are based on academic year:
 - 1st year \$3
 - 2nd year \$8
 - 3rd year \$12
 - 4th year \$15
 - 5th year \$20
 - 2) 3D prints start on first come, first serve basis within the queue.
 - 3) Each student may queue up to 1 job at a time.
 - 4) 1st year students must complete a D2L course provided by the Digital Fabrication Lab before receiving access.
 - 5) 5) The Bambu Lab H2D Pro is only available for 4th and 5th year students.
 - 6) Only CACM class projects are permitted, no personal 3D printing is allowed.
-

Requirements

Listed below are what is required of students

- A computer
 - Access to the internet
 - Your .stl file
 - An account for 3D Printer OS
 - Cura Slicer software
-

Table of Contents

3D Printing Farm Policy	2
Requirements	2
Cura software setup	4 - 11
Slicing a file for 3D printing	12 - 14
Learning the Cura interface	15 - 17
Slicing an object in Cura	17 - 22
Submitting to 3D Printer OS	23 - 28
Approval Process	29 - 31
Troubleshooting	31 - 33

Cura Software Setup

Overview

This section covers how to install and set up Cura for your own personal laptop (Windows).

Cura is already installed and setup in all CACM Labs

Step 1: Downloading Cura

- Go to the [Ultimaker Cura Website](#).
- Scroll down to the latest version
- Select the option under **Windows->Windows Installer (.exe)**



Windows

Windows Installer (.exe) >

> Windows Enterprise Installer (.msi) >
>
>

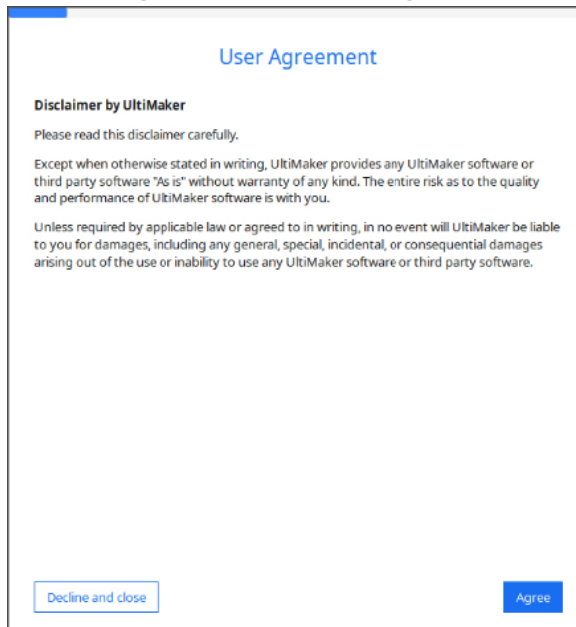
- After the installer downloads, go to your downloads folder->**double click to install Cura.**
- **Select Next** on the setup screen->**select I Agree** for the license agreement.



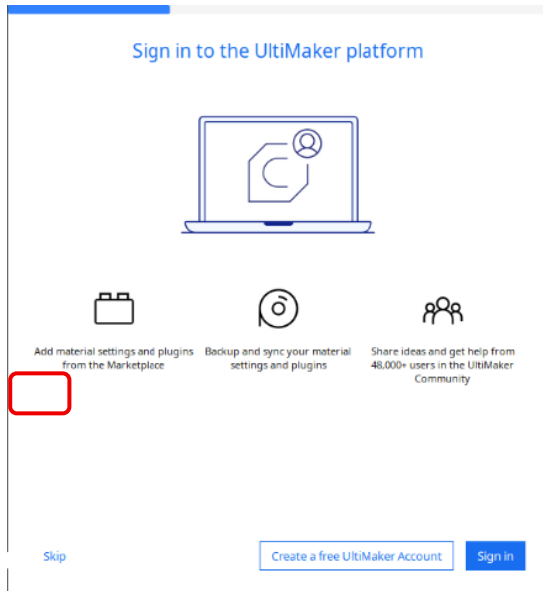
- **Select Install**
 - **Once it's done select finish**
-

Step 2: Setting up Cura

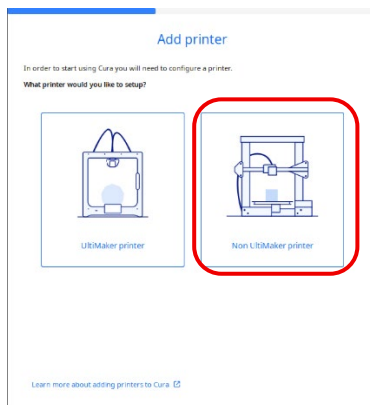
- Open Cura using this shortcut  on your computer
- **Select Agree** on the user agreement



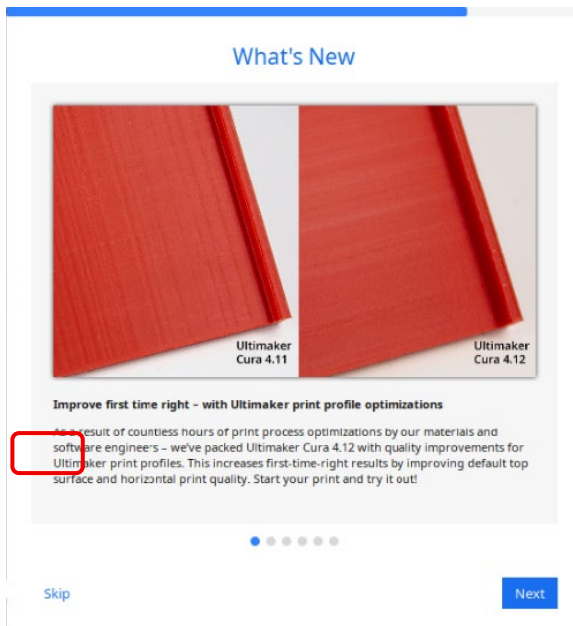
- **Continue selecting agree** until you get to the sign in screen.
- **Select skip** as you do not need an account to use Cura.



- Cura forces you to add a printer before you can proceed, to get past this **select Non Ultimaker Printer.**

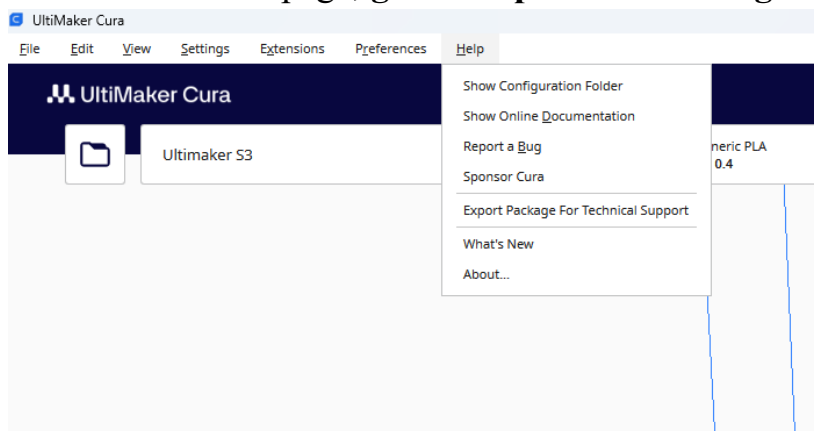


- **Next select add a non-networked printer.**
- **Select Next** to continue.
 - *Note: it does not matter what printer is select because this setting will be overwritten in the next steps.*
- Skip through the next few screens Cura gives you.

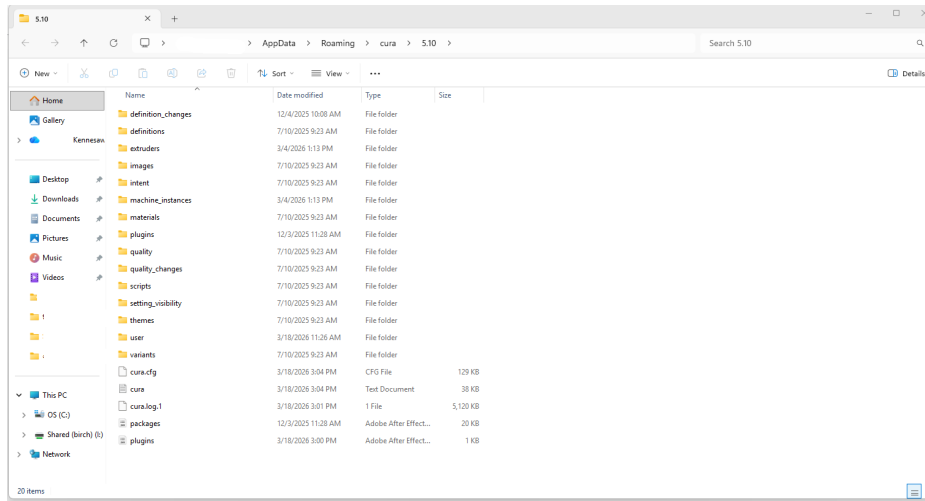


Step 3: Configuring Cura

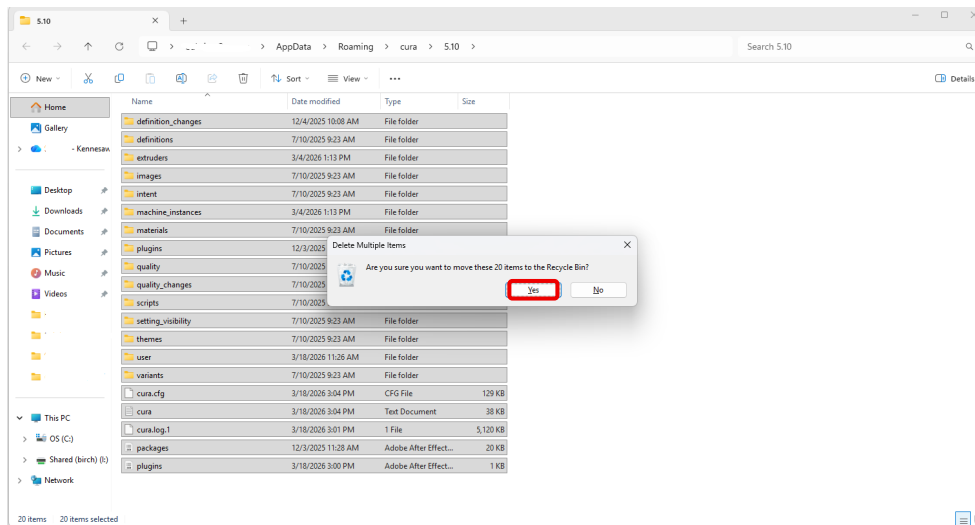
- Once on the main page, go to **Help->Show Configuration Folder**.



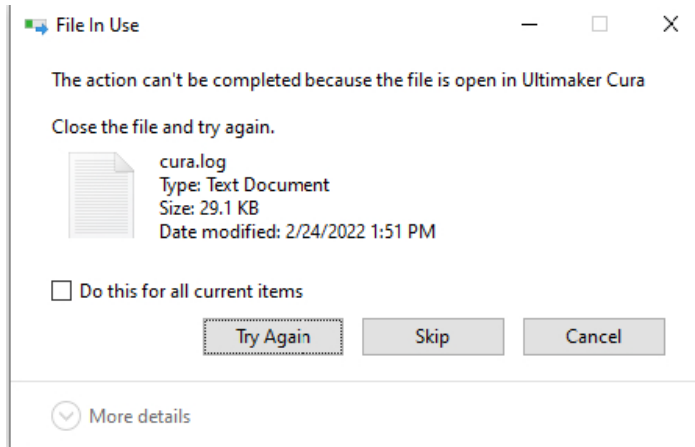
- A configuration folder will pop up.



- **Close Cura, but leave this folder open.**
- **Select all of the files in that folder and delete them.**



- *Note: make sure that Cura is closed or a pop up saying “The action cannot be completed” will appear.*



You do not want to see this pop up!

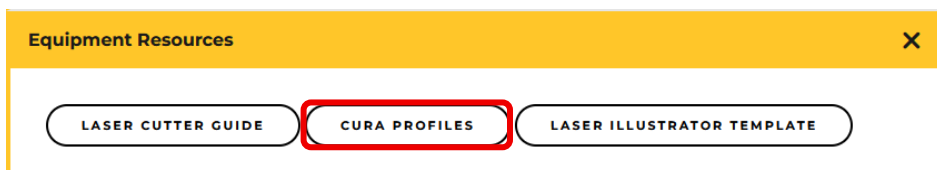
If you have not already downloaded the cura profiles from the Digital Fabrication Lab website, do that now.

Step 3.5: Downloading Cura Profiles

Only do this step if you have not downloaded the Cura profiles from the Digital Fabrication Lab website

If you have already downloaded the profiles skip this step.

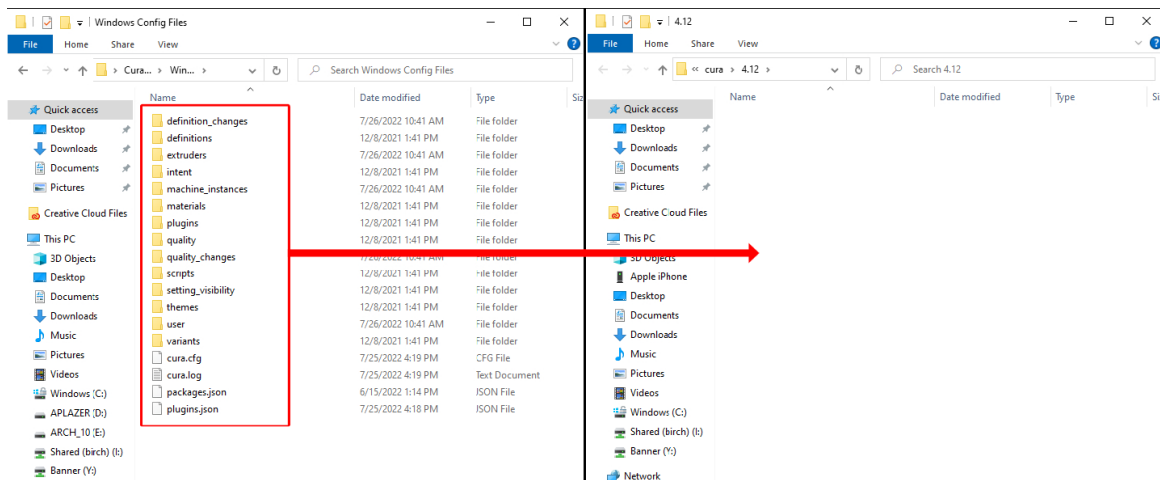
- Go to the [Digital Fabrication Lab website](#).
- Select **Equipment Resources** drop down menu.
- Select **Cura Profiles** from the list of equipment resources.



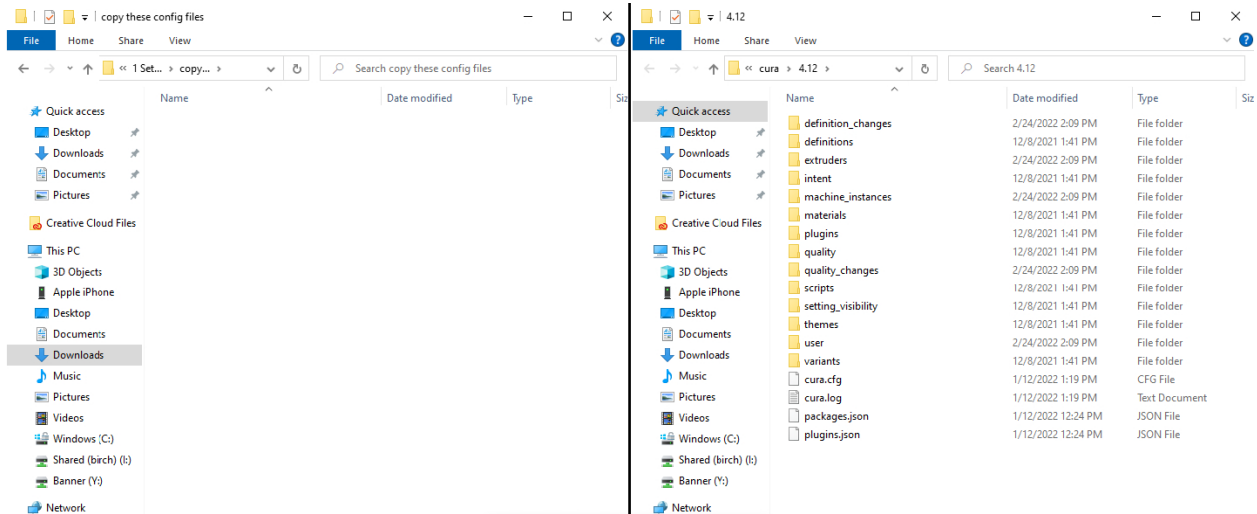
- Go to Downloads→Double click on the zip folder labled **Cura-profiles-windows**→Double click on the folder inside it.
- You can now proceed to the next step.

Step 4: Configuring Your Cura

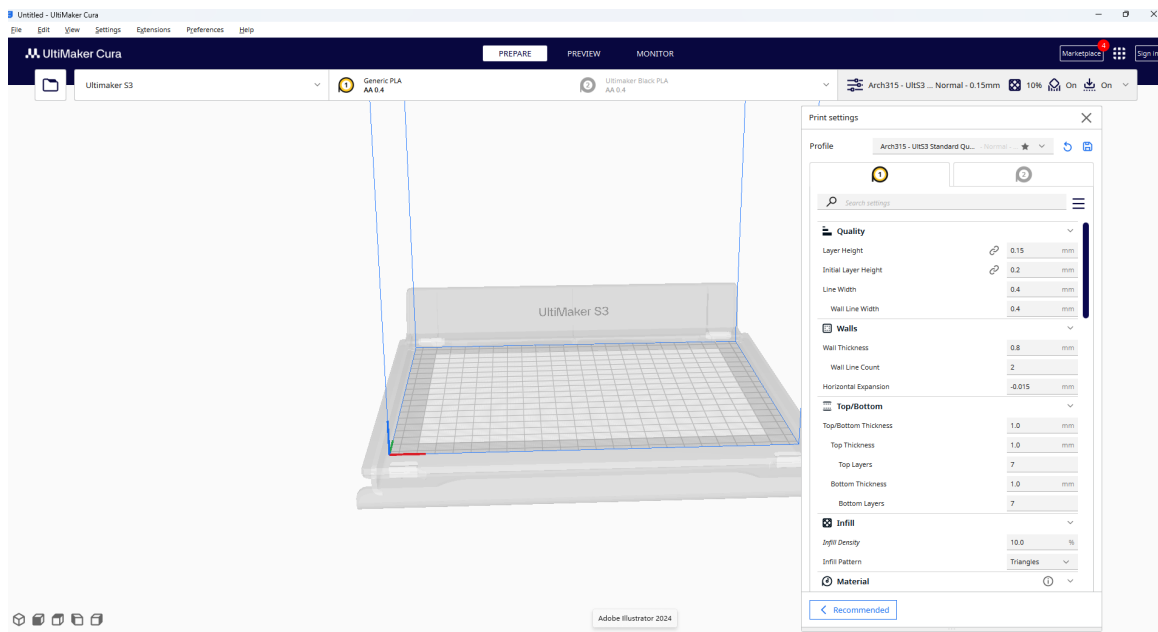
- With the DFL Cura profile folder open, **copy + paste** or **drag all the files inside into the now empty Cura configuration folder.**



- *Note: this will overwrite Cura's settings and add profiles automatically.*
- After moving the files into the Cura configuration folder, close the folders and re-open Cura.



- Cura should now open up with all of the print profiles provided by the Digital Fabrication Lab for the 3D printing farm.
 - *Note: if there is a firmware or beta pop-up at the bottom, just ignore it and exit out of that window.*



- To confirm that your Cura is set up correctly, click on the tab at the top left of the screen labeled Ultimaker S3 next to the 

- In the drop-down menu, there should be an option for **Ultimaker S3**, **Ultimaker S5**, and **Ultimaker S6**.
 - *Note: while your Cura is now fully set up, please be aware that every new academic year (Fall) the Digital Fabrication Lab will update the profiles.*
 - *Note: Students should be mindful and always update their profiles every fall.*

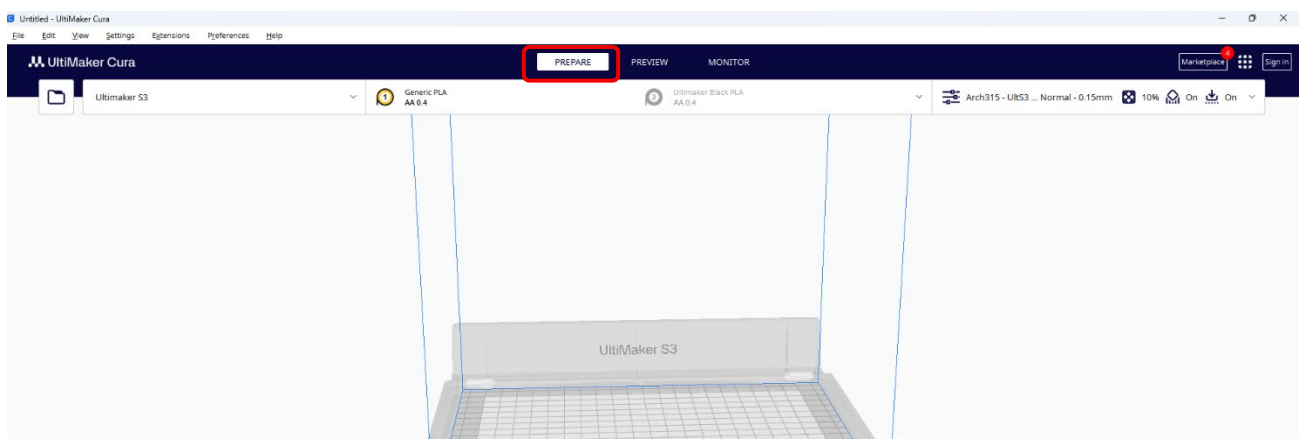
Slicing a File for 3D Printing

Overview

This section will show users how to slice a file using Cura

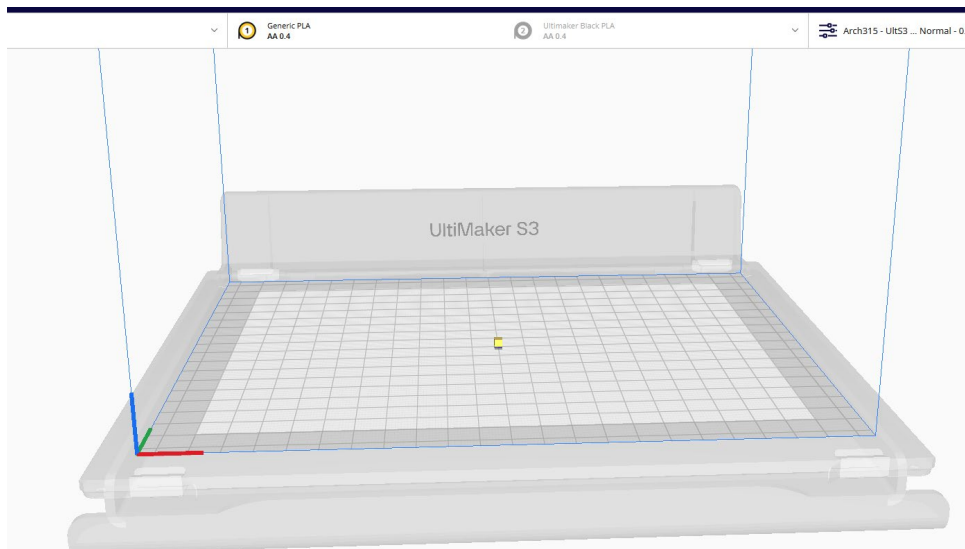
Step 1: Open your file.

- Make sure you are under the **Prepare** tab.



- Go to the printer select tab or the .

- Change it to the 3D Printer you wish to use.
 - *Note: the viewport will change to match the bed size of the 3D printer chosen.*
- Cura will only work in millimeters, this cannot be changed.
 - Make sure the file you are using is converted to millimeters in your original software.
 - If there are multiple files, make sure that millimeters is chosen for every file imported.
- To add a .stl file, you can **drag and drop them into Cura** or go to **File→Open Files→Select the .stl file you need.**
 - If the object imports in very tiny, then the native units were not in millimeters
 - An example would be an object that was supposed to be 3in is now 3mm.

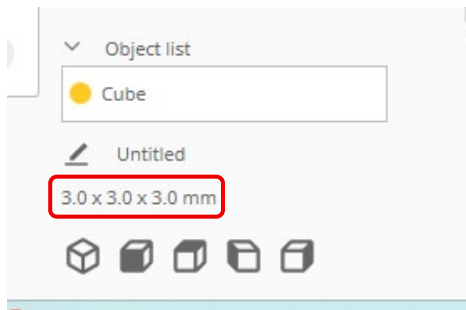


Example of a file that has incorrect units.

If this has happened to you please proceed to the next step, if not please proceed to the next section.

Step 2: Resizing an incorrectly scaled file

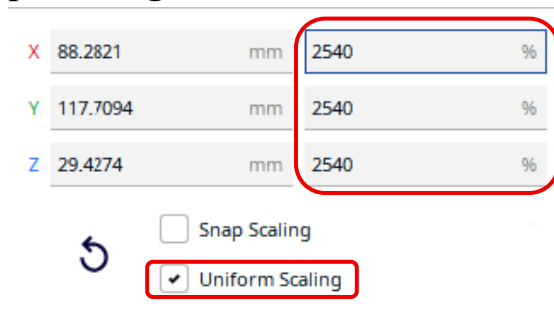
- A way to double check your object size is to look at the measurements on the bottom left-hand corner of the screen. It will tell you what the scale is in millimeters.



- To fix the scale of a file, **make sure the object is selected.**



- **Go to the tool bar on the left-hand side of the screen → scale tool**
- With **Uniform Scaling** selected go to the **Percentages** → **change the percentage to 2540%**



Learning the Cura interface

Overview

This section will show users how to use the interface within Cura to arrange your objects

- On the left side of the screen you will see the ToolBar.
- This is what you will use to move, rotate and scale your objects.



- **Move Tool**, use this to move your object in the X, Y or Z axis.



- **Scale Tool**, use this to scale your object in the X, Y or Z axis.



- You also have the option of **Uniform Scaling** and **Snap Scaling**.

☐ Snap Scaling

☒ Uniform Scaling

- **Rotate Tool**  , use this to rotate your object in the X, Y or Z axis.
 - You also have the option to **Reset**  , **Lay Flat**  , **Select face to align to build plate**  , and **Snap Rotation**.

☒ Snap Rotation

Snap Angle

There are other tools at your disposal to help with setting up your files in Cura.

- Go to **Edit→select it**, listed below are the different types of options.

- Undo
- Redo
- Select All Models
- Arrange All Models
- Multiply Selected
- Delete Selected
- Clear Build Plate
- Reset All Model Positions
- Reset All Model Transformations
- Drop All Models to buildplate
- Group Models
- Merge Models
- Ungroup Models

- **Select All Models**, will have Cura select every individual object on the build plate.
- **Arrange All Models**, will have Cura arrange every individual object so that it's optimally placed on the build plate.
 - *Note: this does not work if all of your objects are concentrated into one file.*

- **Multiply Selected**, will give you the option to multiply a single object.
- **Clear Build Plate**, will remove all objects from the build plate.

Now that you know the navigation tools you are ready to move onto the next section.

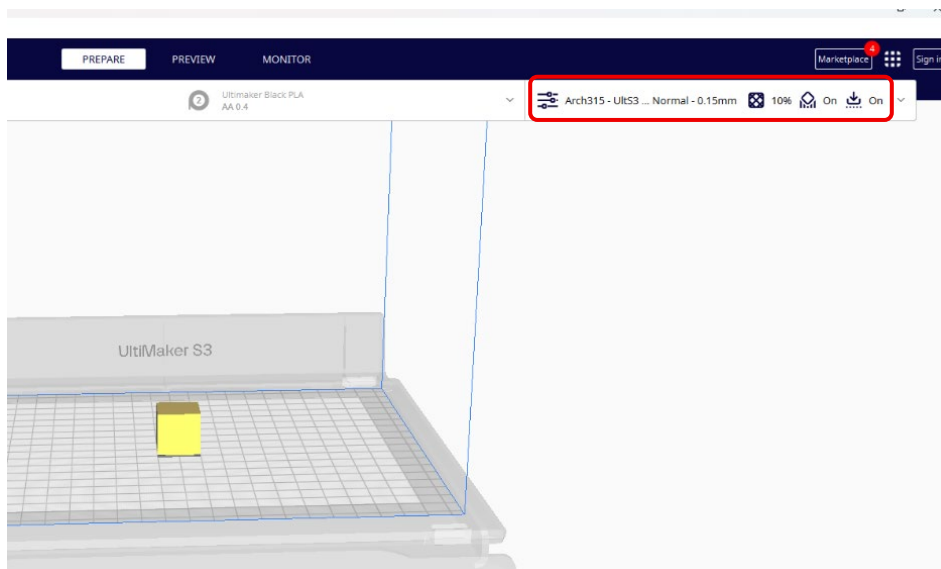
Slicing an object in Cura

Overview

This section will show you how to use print profiles and slice your file within Cura

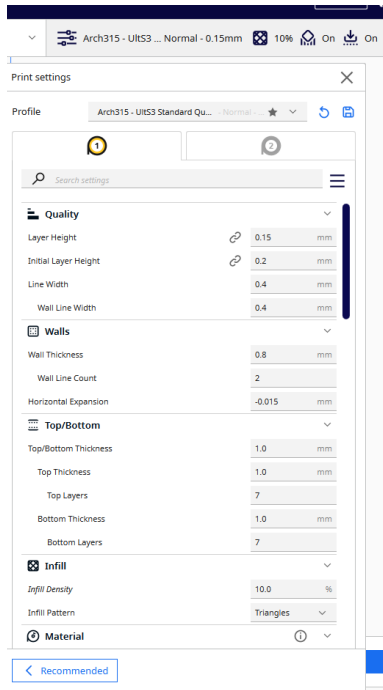
Step 1: Select your print profile

- With your object selected go to the **Print Profile tab** on the right side of the screen.

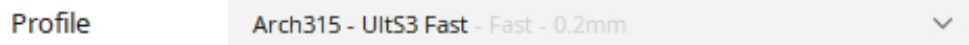




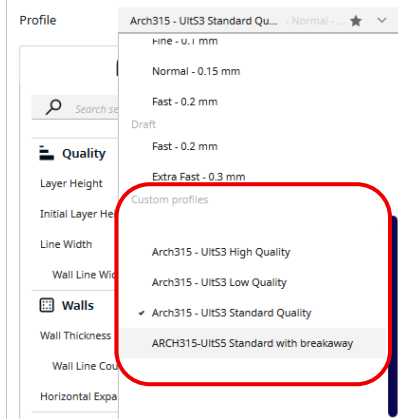
- Clicking on it will bring down the **Print Settings menu**.



- To select the profiles made by the Digital Fabrication Lab select the drop-down menu next to Profile.



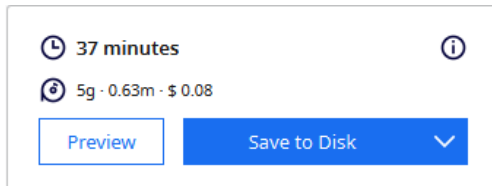
- Scroll down to **Custom Profiles** at the bottom of the menu



- There are different profile settings for different print needs. Pick the one that fits your needs best.
 - *Note: depending on what profile you pick will change the amount of filament used, cost, and how long the print will take.*
 - These settings are already set up so that all you need to do is slice. **Students should not be adjusting these settings.**
-

Step 2: Slice

- Once satisfied, you need to **select Slice**  at the bottom of the screen.
 - Depending on the size of your file, it will take Cura some time to calculate.
- The amount of filament used, cost, and print time will be shown.



Step 3: Double check your work

Before saving the file you want to make sure there are no issues that would cause it to not 3D print correctly.

- To check for this go to the **Preview tab**

PREVIEW

UltiMaker Cura

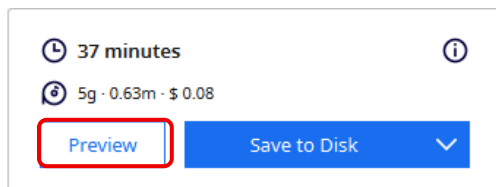
PREPARE

PREVIEW

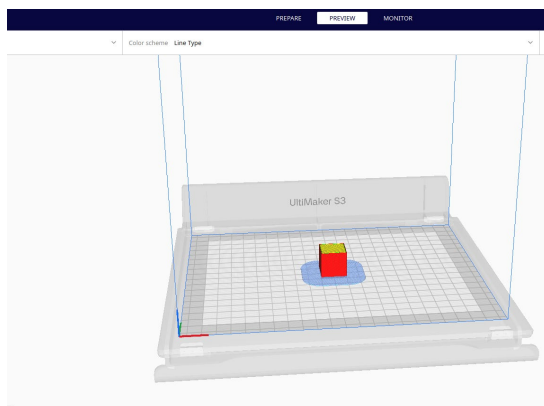
MONITOR

- You can also **select the Preview button** in the same area you sliced your file.

Preview



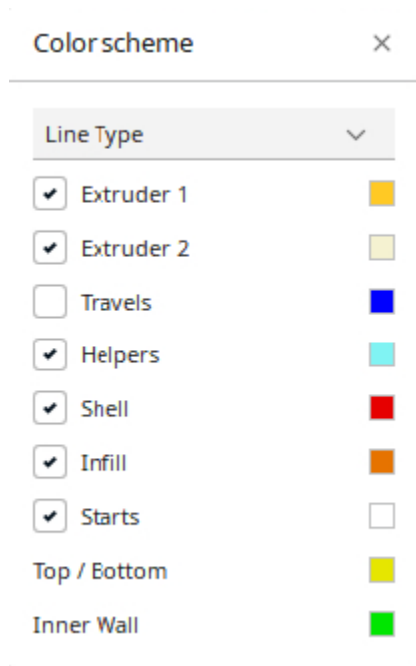
The preview window shows the individual sliced layers of the .stl file.



- At the top are **View Type (Layer View)** and **Color Scheme (Line Type)**.



- **Select Color Scheme (Line Type)** to discover what the various colors on your object mean.



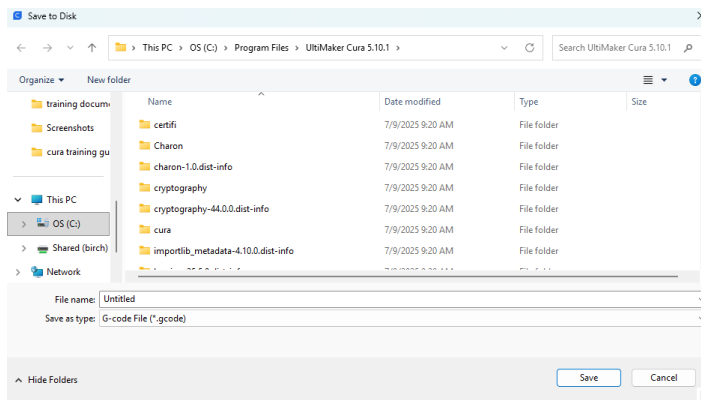
Here are definitions for the most common colors you will encounter.

-  **Helpers:** Are the support and build plate adhesion
-  **Shell:** The outer layer of the 3D print.
-  **Infill:** The inner structure of the 3D print.
-  **Top/Bottom:** The top and bottom layers of a 3D print.
-  **Inner Wall:** The wall between the infill and top/bottom layers.

- If you spot any abnormalities, then there may be issues with the mesh of the imported object.
 - *Note: Meshes cannot be fixed in Cura, so the best option is to return to the original software to address any issues and try again.*

- **Select Save to Disk.**
- Save your file to your computer, make sure to change the file type from. ufp to .gcode.

You must make sure that you change the file type too. Gcode as that is the only file type 3D Print OS takes.



Submitting to 3D Printer OS

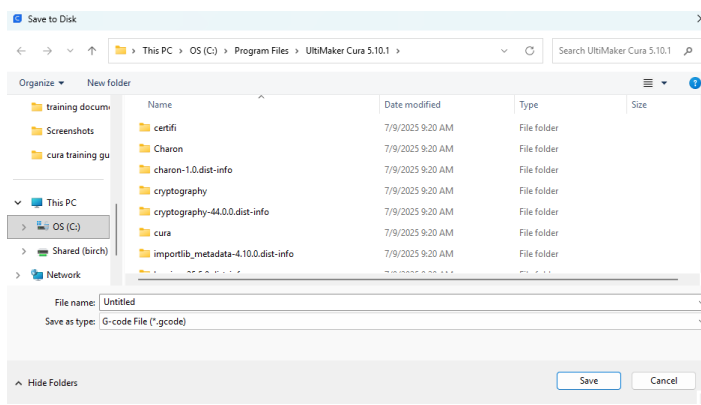
Overview

This section will show you how to upload your .gcode file to 3D Printer OS

Step 1: Save your .Gcode file

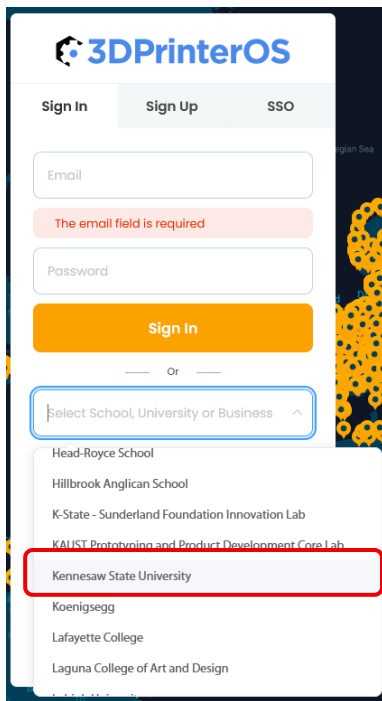
- Select Save to Disk.
- Save your file to your computer, make sure to change the file type from .ufp to .gcode.

You must make sure that you change the file type too. Gcode as that is the only file type 3D Print OS takes.

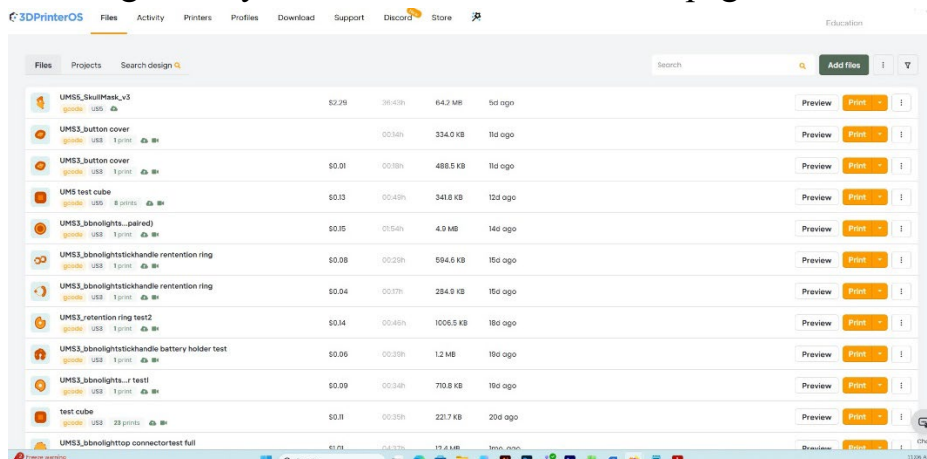


Step 2: Log into 3D Printer OS

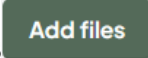
- Using single sign-on log into [3D Printer OS](#)

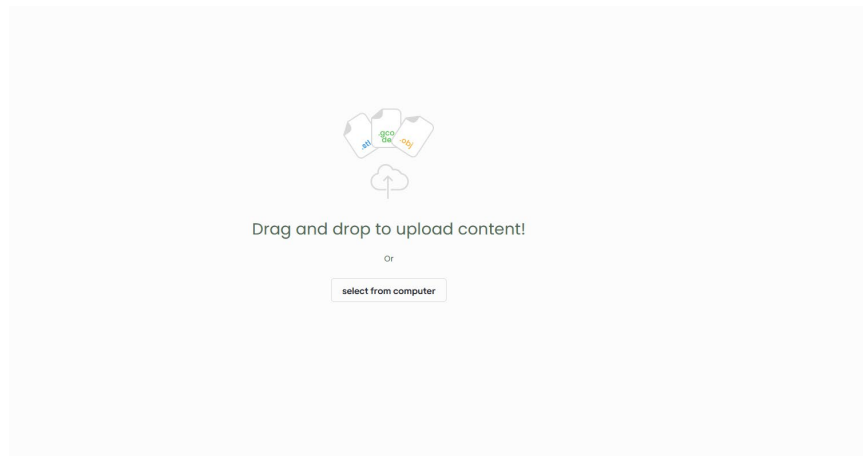


- Once signed in you will be taken to the **Files** page.

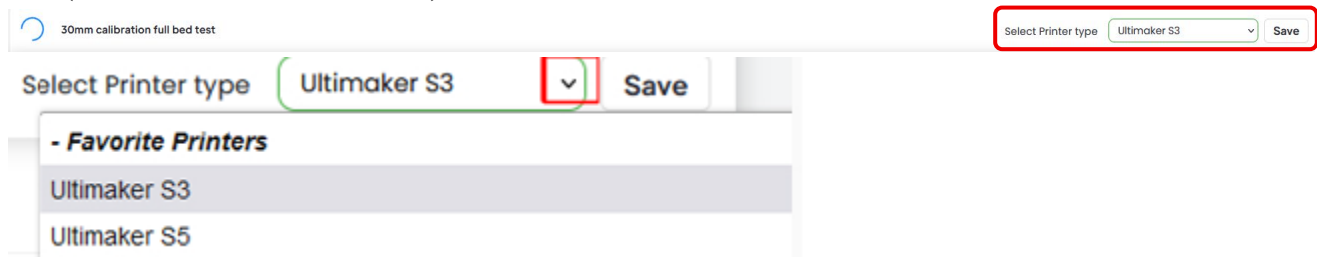


Step 3: Upload the .gcode file

- Select the **Add Files**  button.
 - You will be given the option to **Select from Computer** or **Drag and Drop** your .gcode file in.



- Add your newly created .gcode file.
- Go to **select Printer Type** → **select the 3D Printer you want to use from the drop down menu**. This depends on what 3D printer you sliced the file for (Ultimaker S3, S5, or S6).



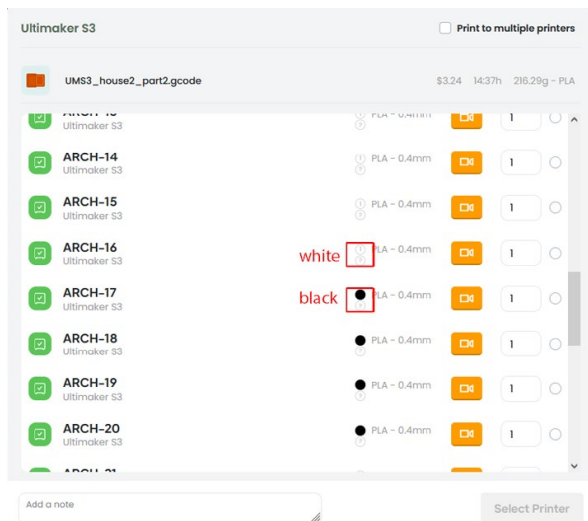
- *Note: if the incorrect 3D printer is selected, your print will be rejected once sent to the queue.*

- Select **Save** 

Step 4: Select your 3D Printer

Once the object is fully uploaded, the Preview and Print options will appear.

- A list of available 3D printers will appear
 - Only 3D printers compatible with the ones you selected will appear.
 - *Note: when choosing a machine, take note of filament color. Most machines will have white PLA but there are a few with black.*



The colors on the 3D printing icon will indicate the status of that 3D Printer



The 3D Printer is available for use.



The 3D Printer is currently not available.



The current 3D print job is paused.



The 3D printer is currently being used.

- If a 3D printer has one or multiple jobs queued or printing on it, students will be able to see a projected time of how long the wait is.

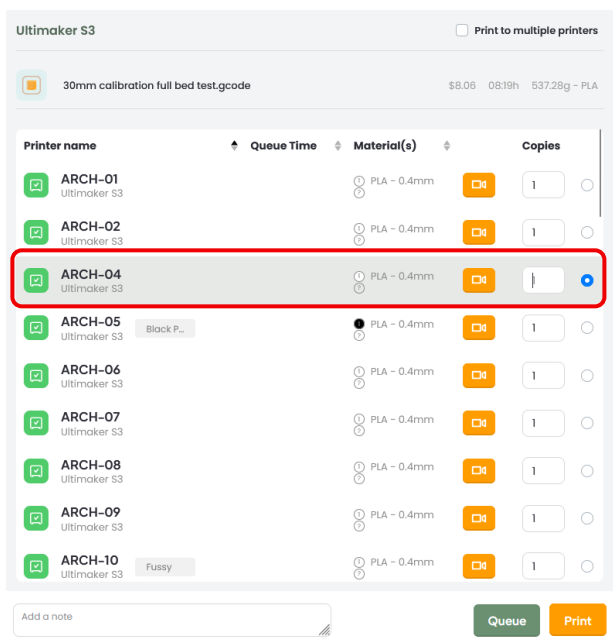


ARCH-08
Ultimaker S3

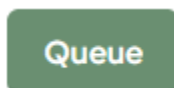
16:18h

During peak traffic season Midterms and Finals, the wait time for 3D printers can increase dramatically. Please keep this in mind when creating your projects.

- Select the 3D printer you want by **selecting the radio button.**

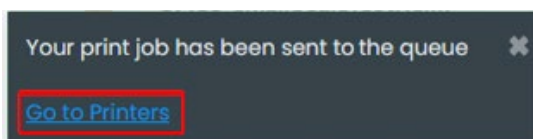


- With your 3D printer selected you can now select the Queue button



- *Note: students will only be able to queue their prints*

You will receive a pop up that the print has been queued to the 3D printer.

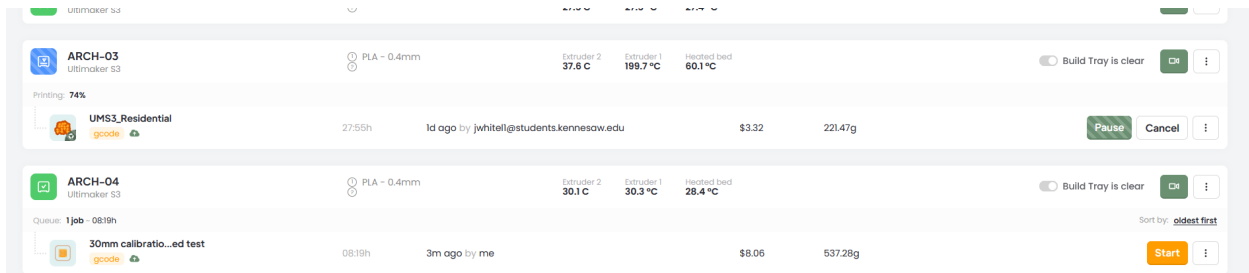


- Select **Go to Printers** link on the pop-up, or you can **select the Printers** tab at the top of the screen.

Step 5: Find your file

The Printers page shows the progress and queue of all machines in the 3D printing farm.

This is where you will find your recently submitted print job.



The screenshot displays the 3DPrinterOS interface for two Ultimaker S3 printers, ARCH-03 and ARCH-04. Both are using PLA - 0.4mm filament. ARCH-03 is currently printing a file named 'UMS3_Residential' at 74% completion, with a duration of 27:55h, submitted 1d ago by jwhite@students.kennesaw.edu. ARCH-04 is in the queue with a job named '30mm calibratio...ed test', submitted 3m ago by me. The interface shows temperature readings for extruders and the heated bed, as well as cost and weight information for each job.

Printer	Job Name	Material	Status	Duration	Submitted	Cost	Weight
ARCH-03	UMS3_Residential	PLA - 0.4mm	Printing: 74%	27:55h	1d ago by jwhite@students.kennesaw.edu	\$3.32	221.47g
ARCH-04	30mm calibratio...ed test	PLA - 0.4mm	Queue: 1 job - 08:19h	08:19h	3m ago by me	\$8.06	537.28g

Approval Process

Overview

This section will explain the approval process for the 3D Print farm

3D print job approvals

- Your 3D print job will have to go through a vetting process by the staff at the Digital Fabrication Lab.
- Staff are specifically looking for any abnormalities that would cause a 3D print job to fail or cause damage to the machines.
 - *Note: lab staff will start the print, usually within 1 hour of submission during the lab's operational hours.*
 - *Note: Print jobs submitted after hours will be reviewed on the following business day*
- Students will not specifically get an email from the Digital Fabrication Lab about their file. **All emails will come from 3D Printer OS.**

Students will receive an email once a print job is started, stopped, or completed.

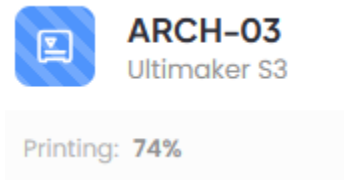


Checking a print job's progress

Students can check the progress of their Print job through the Printers page on 3D Printer OS.

There are several different ways to check progress:

- **Printing percentage** shows how long before the print job is completed.

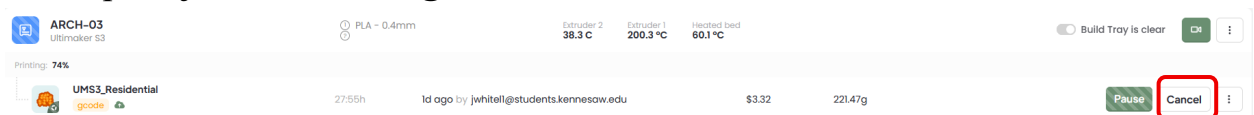


- **Live Camera** all 3D printers come with a live feed of their print job.
- To access it select the Camera button  on your active print job.

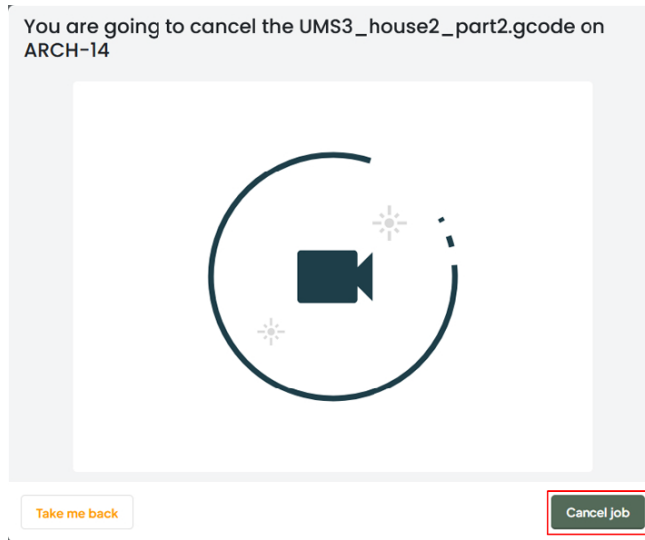


Canceling a Print Job

- If you wish to cancel a current print job, you can do this by going to your active print job and **selecting the Cancel button**  .



- A confirmation window will pop up, **select Cancel Job** to confirm.



If a student actively cancels their own print job, they will not be refunded the amount of money used.

If a student's print job is cancelled due to an error on a student's part:

- No support when needed
- No build plate adhesion when needed
- Design flaw causing it to fall apart
- Thin walls

The print will not be refunded if cancelled by staff.

Refunds are only issued to a student's print job if it's stopped due to mechanical error.

Troubleshooting

Overview

This section will explain the most common errors students will run into when using Cura.

Cura Specific:

Problem	Cause	Solution
File comes in too small.	File was not scaled correctly in original software.	Select the object→go to Scale on the toolbar→change the percentage to 2540%
Objects are floating above the build plate.	Student imported a file without checking that all objects were on the same plane.	- If all of the parts are within one file, the student will need to make the adjustment back in the original software. - All parts need to be on the same level to be flat to the build plate.
There is no support when needed.	Student did not use the right Print Profiles or turned off Support.	- Go to print profiles→scroll down to Support→check it on. - Go to Custom Profiles→make sure you are using the profiles provided by the DFL.
Walls or parts on my object are transparent gray when previewing	- Student has walls that are too thin to print successfully - Student has imported a mesh that cannot be printed.	Go back to the original software the file was imported from, this is a problem within the mesh and cannot be fixed in Cura.



Cannot fit object on build plate with RAFT enabled	Student has an object that is too big to print with baseline Build Plate Adhesion settings.	• Change to BRIM under Build Plate Adhesion settings under Print Profile.
Object is too large to print on an Ultimaker S3	Student has imported an object that is larger than the build plate.	• Re-slice for an Ultimaker S5, if available. • Student will either need to adjust scaling. • Adjust the orientation of the model to fit the build plate. • Slice the model in half and submit two print jobs.

3D Printer OS Specific:

Problem	Cause	Solution
No available 3D printers when trying to queue.	• Selected the wrong 3D printer when submitting .gcode file. • Not added to workgroup.	• Double check the selected 3D printer on your file. • Email staff about being added to workgroup.
Cannot log-in because there is no account.	• User has not been signed up for 3D Printer OS.	• Email staff about requirements to be added.
There is only an option for layout or slice.	• Student uploaded an .stl file instead of a .gcode.	• Submit the correct .gcode file. • Use the 3D Printer OS Slicer.
There is no more money in my 3D Print OS account.	• Student used their given allotment.	• Students have the option to talk to their faculty member and email the chair if more funds are needed.

