How to 3D Print at the SHED Makerspace

Using 3DPrinterOS for Personal, Club, and Class Projects

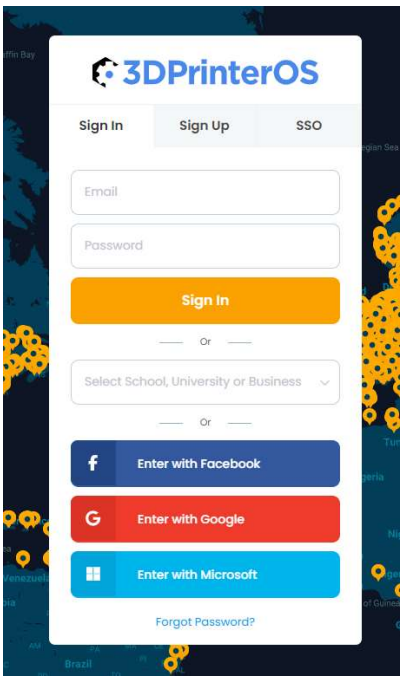
January 2024

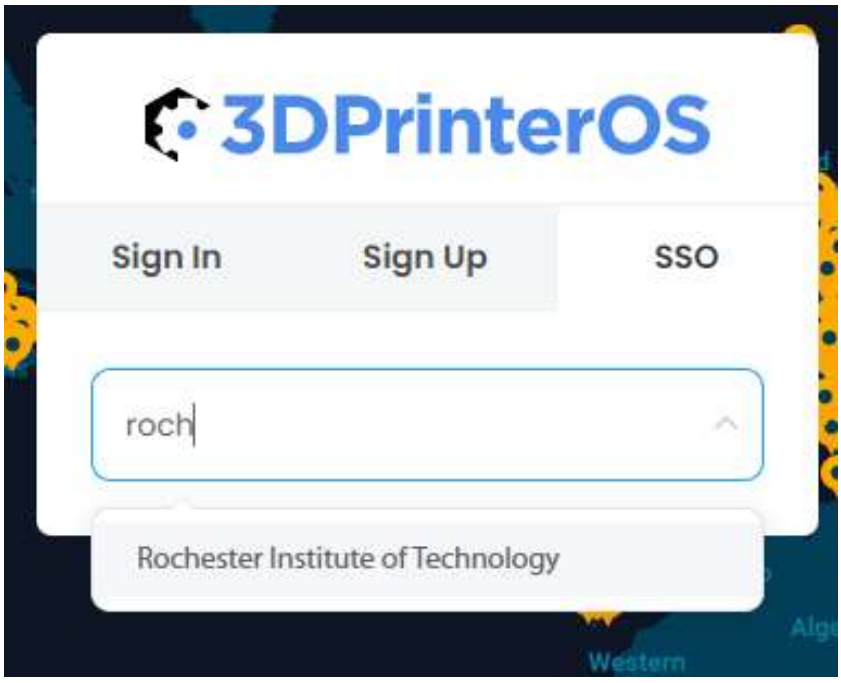
Overview

The SHED Makerspace is excited to announce we are now using 3DPrinterOS to manage our 3D printers. This will make it easier and faster for students, faculty, and staff to 3D print! You can now submit 3D prints from any network-enabled device in the world, 100% through your browser. Here's how;

Logging In

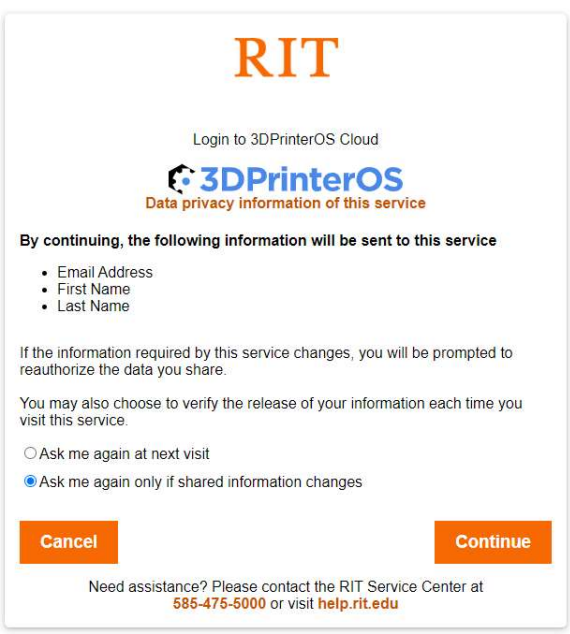
3DPrinterOS can be accessed by going to cloud.3dprinteros.com. From there, press "SSO", and either type in "Rochester Institute of Technology" or select it from the drop down, as seen below. Selecting "Rochester Institute of Technology" will direct you to the SSO page you are familiar with from other on-campus resources, where you can log in like normal with your RIT credentials. This may require Duo 2-factor authentication.





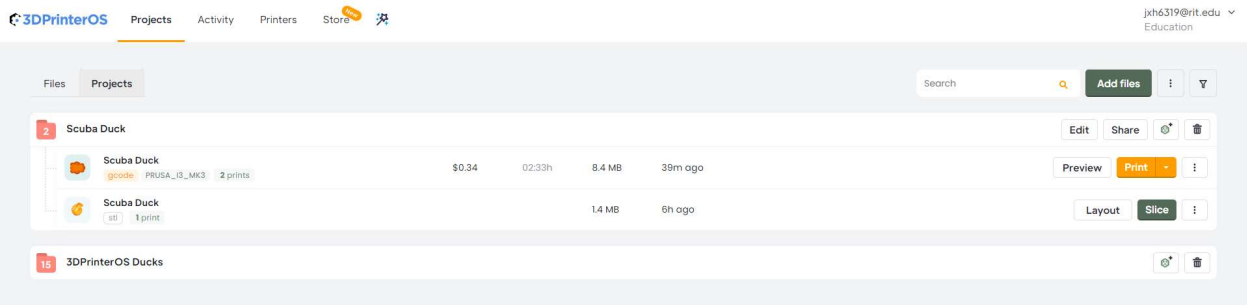
Once you have entered your RIT credentials, the first time you login you will have to go through 2 menus, seen below. The first is a confirmation of what information will be shared from RIT to 3DPrinterOS (name, email). It is suggested you choose "Ask me again only if shared information changes" to avoid having to go through this every time you log in. The second menu is to agree to 3DPrinterOS's terms of service. You can simply select the top check mark to agree to all of them, or

choose to select only certain ones to agree with. Some are optional to use 3DPrinterOS, some are mandatory. Once ready, you can click “Confirm” to access 3DPrinterOS!

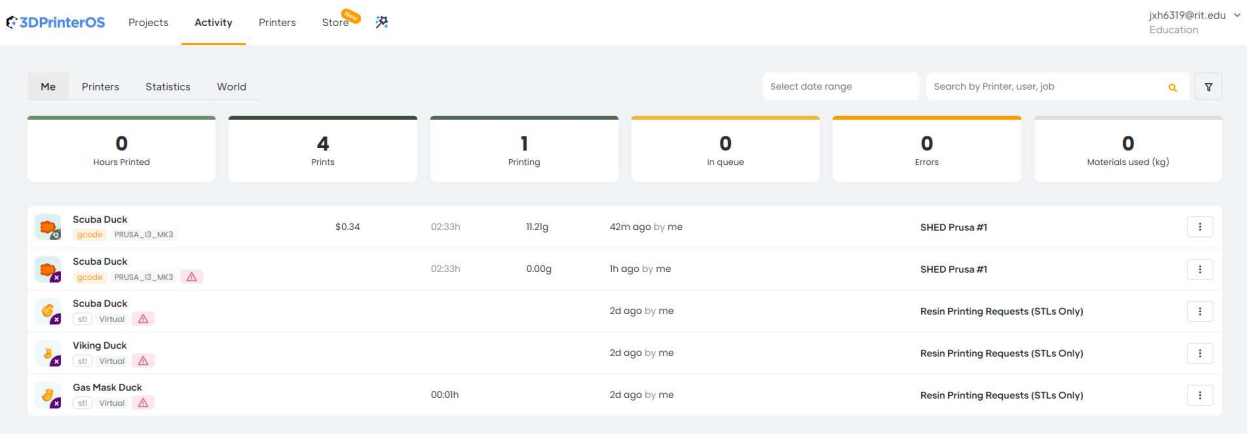


Navigating 3DPrinterOS

When you log in to 3DPrinterOS, by default you will be in the “Projects” tab, seen below. This is where your data is stored. 3D models can be uploaded using the “Add files” button on the right. When files are added, they will appear in the “Files” tab. By default, you will have a number of rubber duck example files.

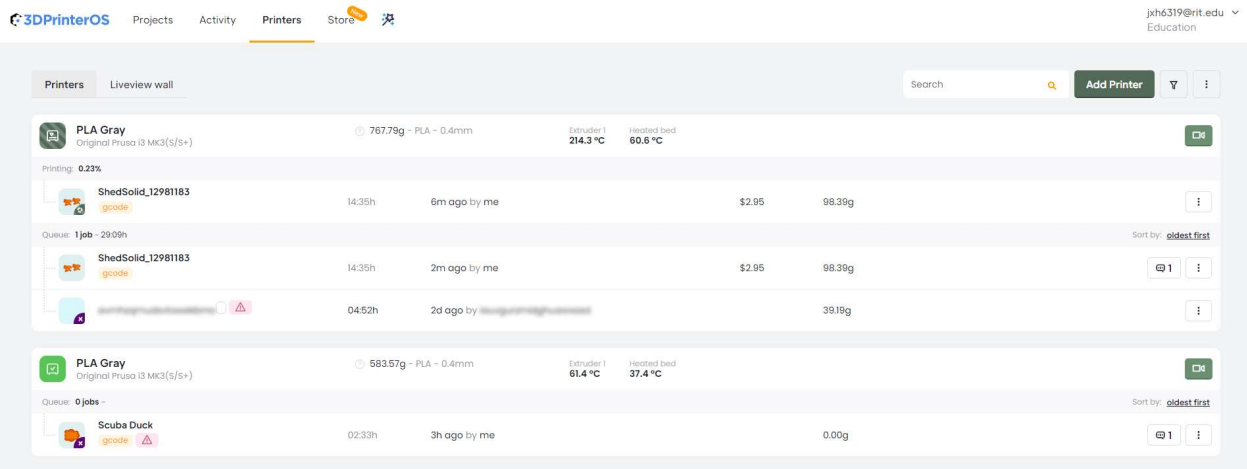


The “Activity” tab is the next one on the top. In the activity section, you can see statistics about your past print jobs, as well as all RIT prints and/or all prints around the world using 3DPrinterOS. This is a useful spot to check the prices of your prints, error messages, and the state of prints (completed/queued/canceled/etc.). You can click on a job to see more information, including a time-lapse video of the print. If you see a red indicator next to a print, you can hover over it to see an error message or warning about your print.



The third tab on the top is the “Printers” tab, where you can see all 3D printers you have access to. From here, you can see what printers are or are not running by hovering over their icon, see how much filament they have remaining, and press the green icon to bring up a live feed of the printer. Any prints in queue or currently printing that belong to you can be seen here. Prints by other members of the RIT community are visible to show something is running/queued, but the name of the print, who it belongs to, and what it looks like are all hidden.

You can also press the “Liveview Wall” button to see the live feeds from all printers at once!



The other 2 buttons along the top of the program – Store and the magic wand (feature request) are not relevant to regular users and can be ignored. You can click on your email address in the top right to sign out when done using the software on a shared or public computer.

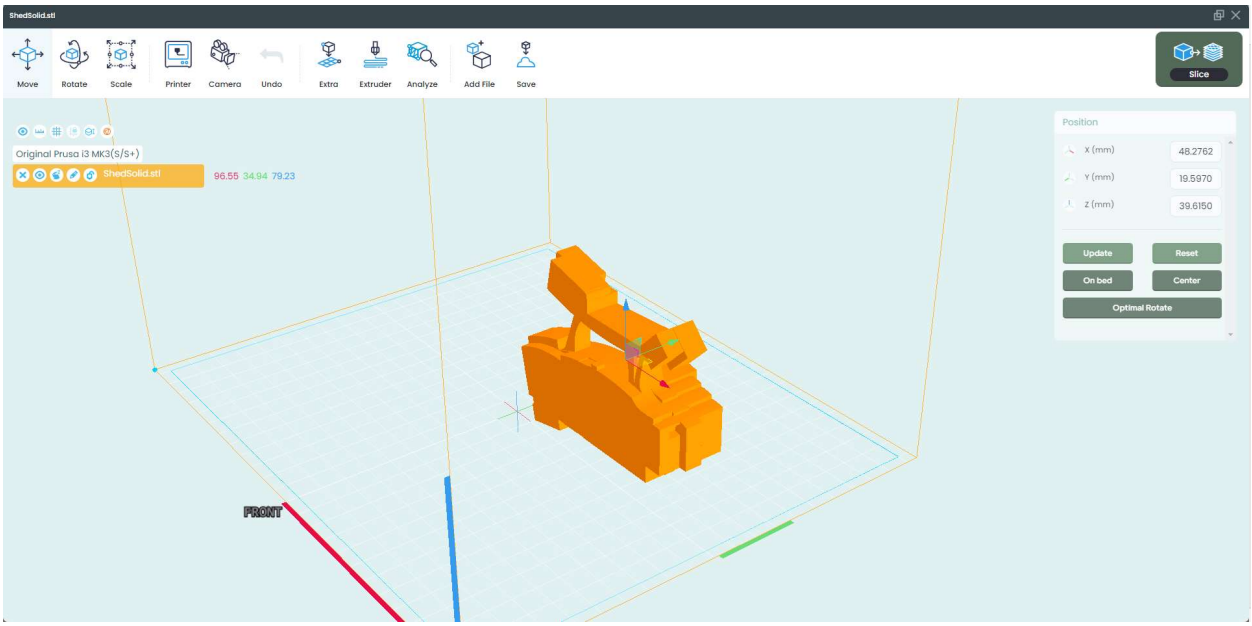
Using 3DPrinterOS to Print

To start a print using 3DPrinterOS, first upload a file by pressing “Add files” under the “Projects” tab. For this example, we will print a scale model of the SHED.

STL, OBJ, 3MF, or similar mesh files are preferred for 3D printing, but 3DPrinterOS can process many different file types. For instance, many CAD files like STEP or IGES will also work. When these are uploaded, you will not have the same “Layout” and “Slice” options seen below, and will first have to select “Convert” in the “…” dropdown to use the Magic Fix tool to convert it to a mesh file.

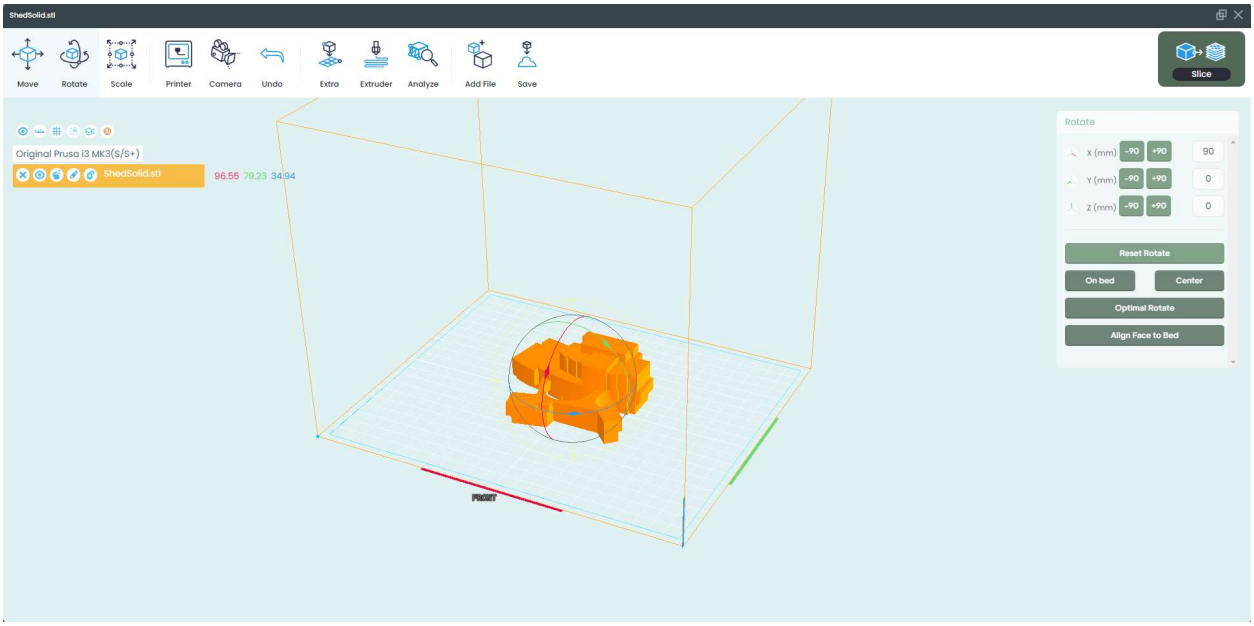
Once your mesh 3D file is uploaded, it will appear in the “Files” tab.

From here we can choose to slice the file directly, but it is recommended first to make sure it is oriented and scaled properly in the “Layout” menu.

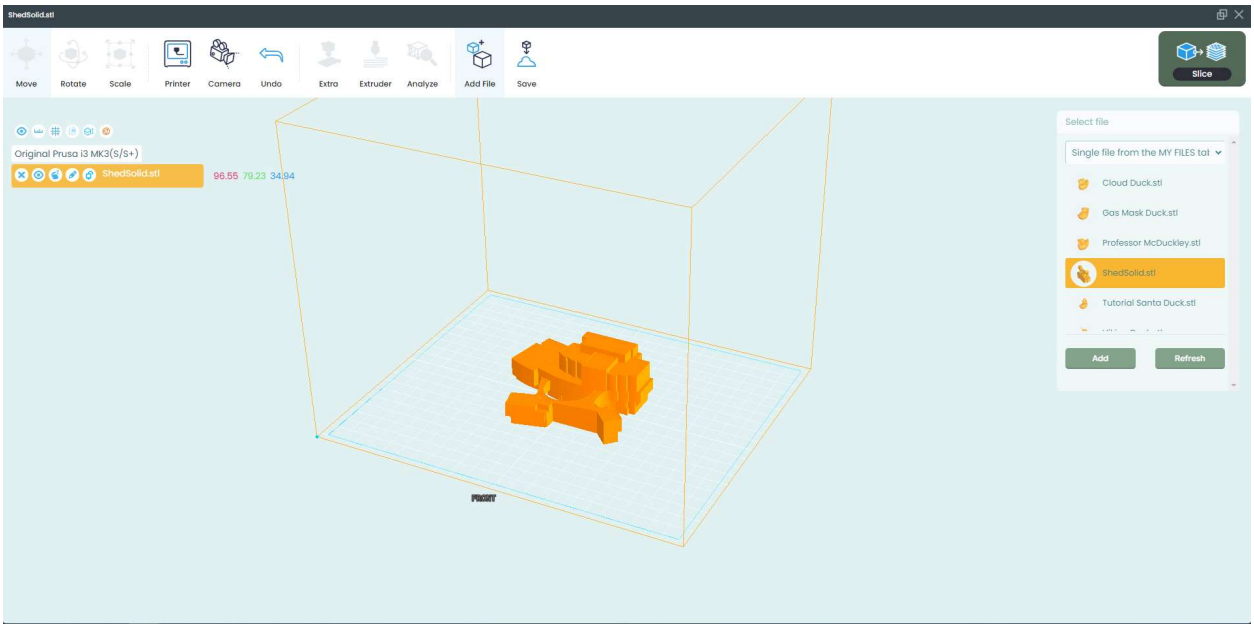


Pressing the “Layout” button will bring up a rendering of our 3D model, placed in a cube the size of the printer’s bed. The first thing we should check is that the scale is correct. On the left side, we can see the X, Y, and Z dimensions in millimeters written in red, green, and blue respectively. If this is not correct, we can use the “Scale” tool at the top to adjust the size. You can then enter either new dimensions in millimeters or scale percentages on the right. You will also find buttons to convert a file from inches to millimeters, or scale it to as large as possible.

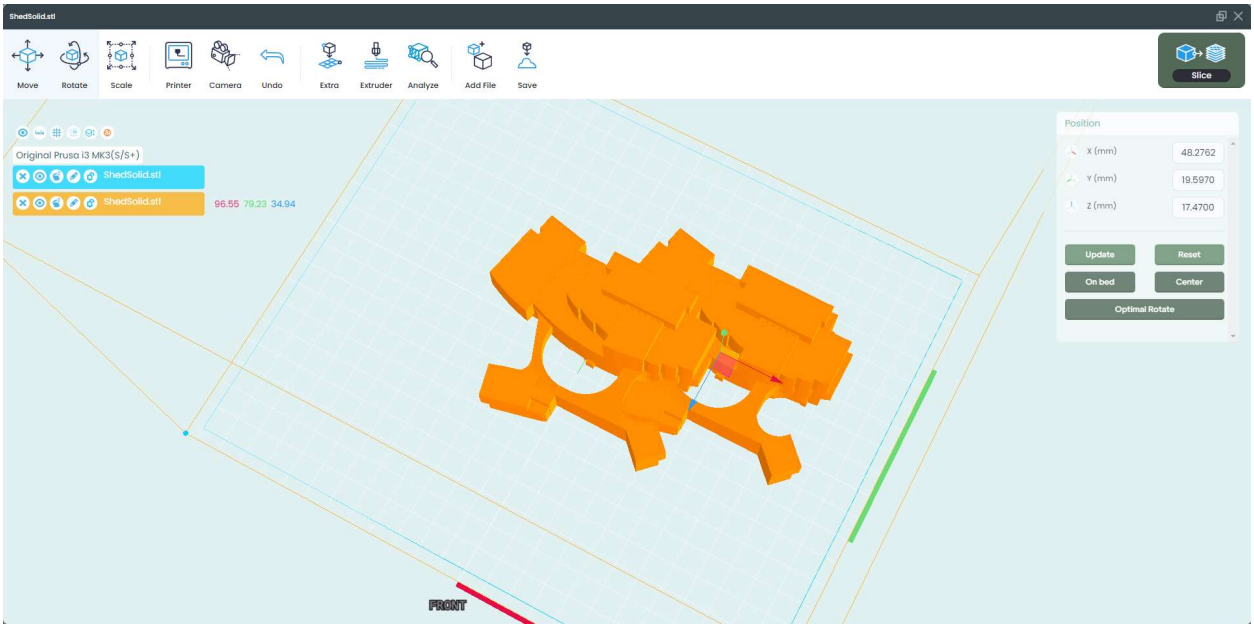
Once we have scaled our object, the next most important thing is to rotate it. We want as much of the part flat against the bed (the base of the printer) as possible. This increases the odds of success, and reduces the amount of support material your print will need. When the 3D model is selected (select it by clicking on it) on the right you can press “Optimal Rotate” at any time to try and align the part to the bed. If you want a different orientation, you can select the “Rotate” tool on the top and then manually enter degrees of rotation on the right. In our case, the software seems to have chosen a good orientation!



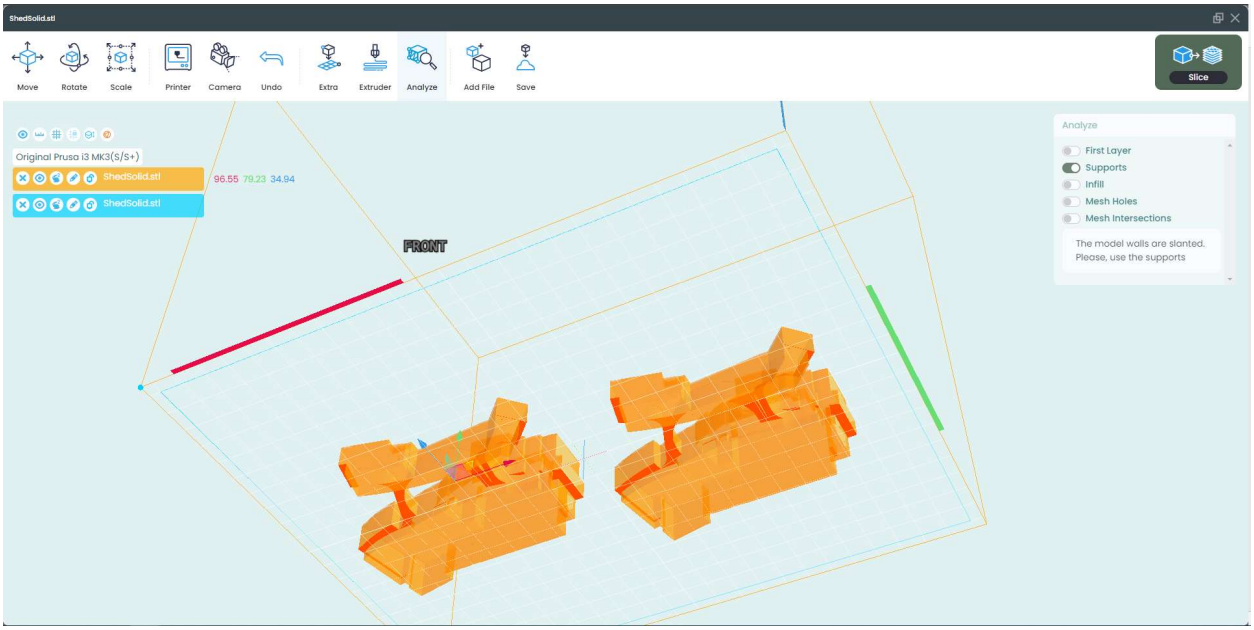
Printing parts one at a time is recommended, but you can add more parts to the bed by pressing “Add File” on the top menu. This will then let you select another file already uploaded to 3DPrinterOS to load. For example, I am going to load another SHED model. Make sure to go through the scale/orientation steps again, and make sure all the 3D models fit on the bed. When you have multiple models on the bed, click on it to indicate which one should be selected for rotating/moving/etc. You can also select them in the list on the right.



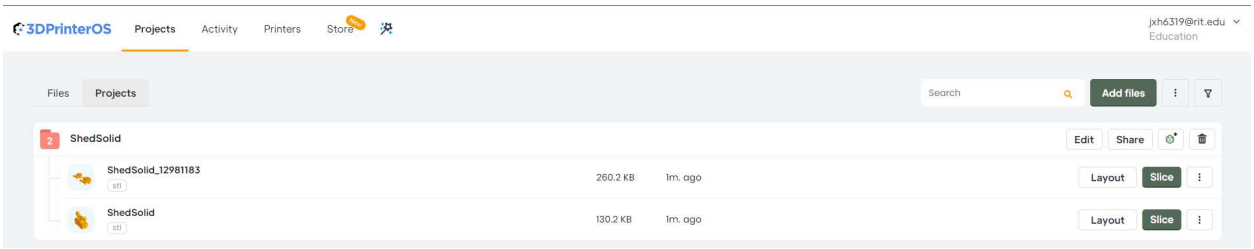
When loading multiple models, also keep in mind that they may be overlapping. Use the "Move" tool to move around the models and make sure they are not overlapping. You can either enter an absolute position on the right, or click and drag on one of the planes/arrows that appear on the model to move it around.



Once you have the models arranged, let's check what the computer thinks. Pressing "Analyze" at the top will pop up a window on the right where we can analyze different aspects of the print. For instance, if I select "Supports", the software will tell me I likely need supports and show me where on the 3D models I will likely need them.

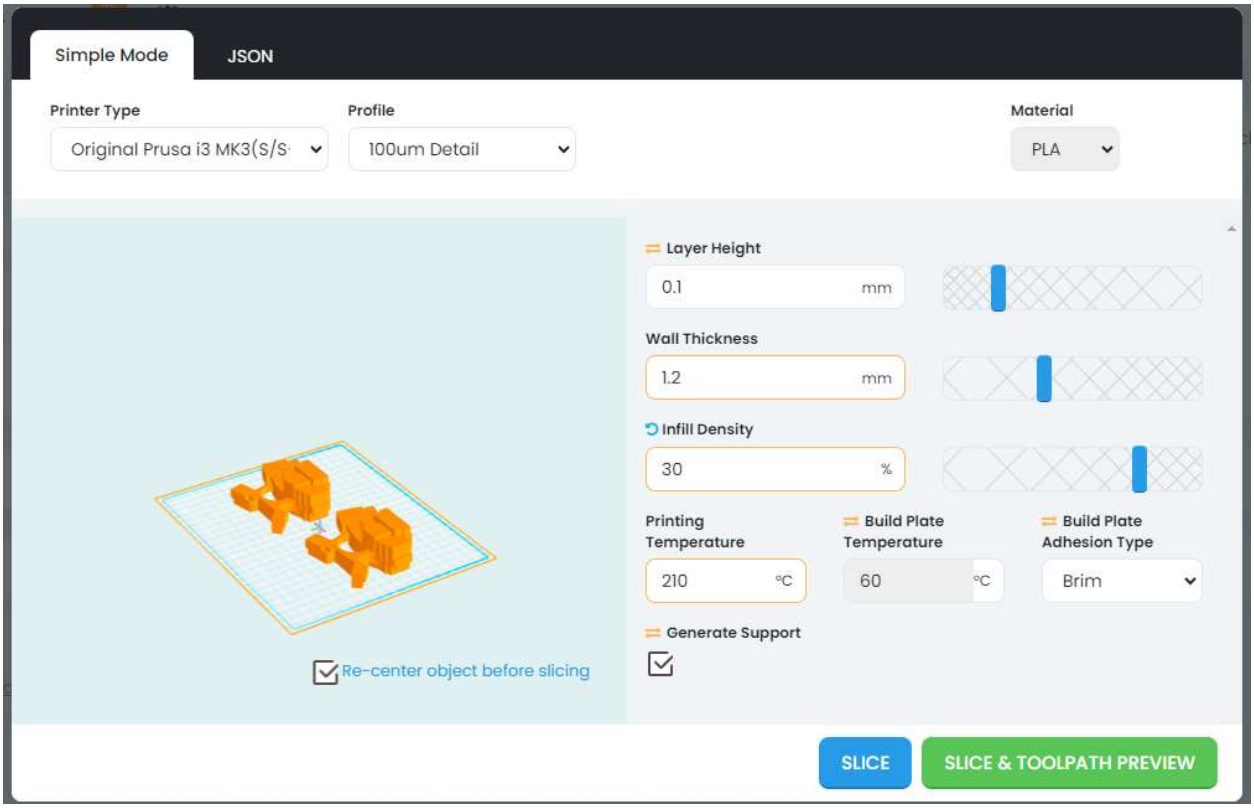


Once everything is set, we can move on to slicing! Slicing is the process of turning 3D models into the actual machine code the 3D printer reads. You can either slice now by pressing the “Slice” button in the top right, or you can save everything and slice later by pressing “Save” at the top. When you save or slice, the file will be turned into a new project, found under the “Project” tab. If I were to save, I would see them like below. The project will track what original STL was used, then have a newer STL representing the arrangement and orientations we set. I want to make sure to select “Slice” on the newer (top) option of these two. Once you slice a file, you can also see that sliced G-code here.

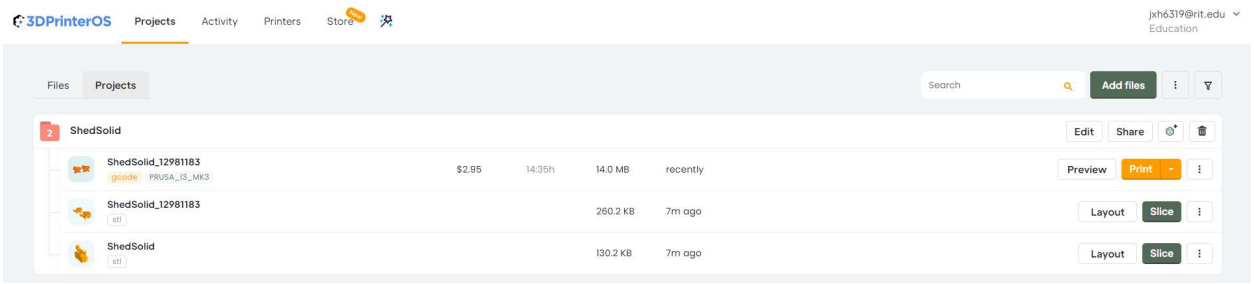


The slicer in 3DPrinterOS is very easy to use. On the top we can select what printer type we are printing on (as of writing, the makerspace only has 1 type of printer available through 3DPrinterOS), as well as pre-made profile. The profile determines the default settings used for a print, but you can tweak some of them suing the sliders on the right. The number in each profile’s name refers to the layer height. Smaller layers take longer to print and are weaker, but look much nicer. Since I want these models to look nice, I will do them at “100um Detail”. But, maybe I also want the prints to be stronger, so I will increase the default 15% inflll up to 30%. I will also make sure to keep support materials turned on, since the analyzer said I need them.

More experienced 3D printer users can request access to all Cura slicer settings to fine-tune their prints. Please speak to a maker mentor for more details.

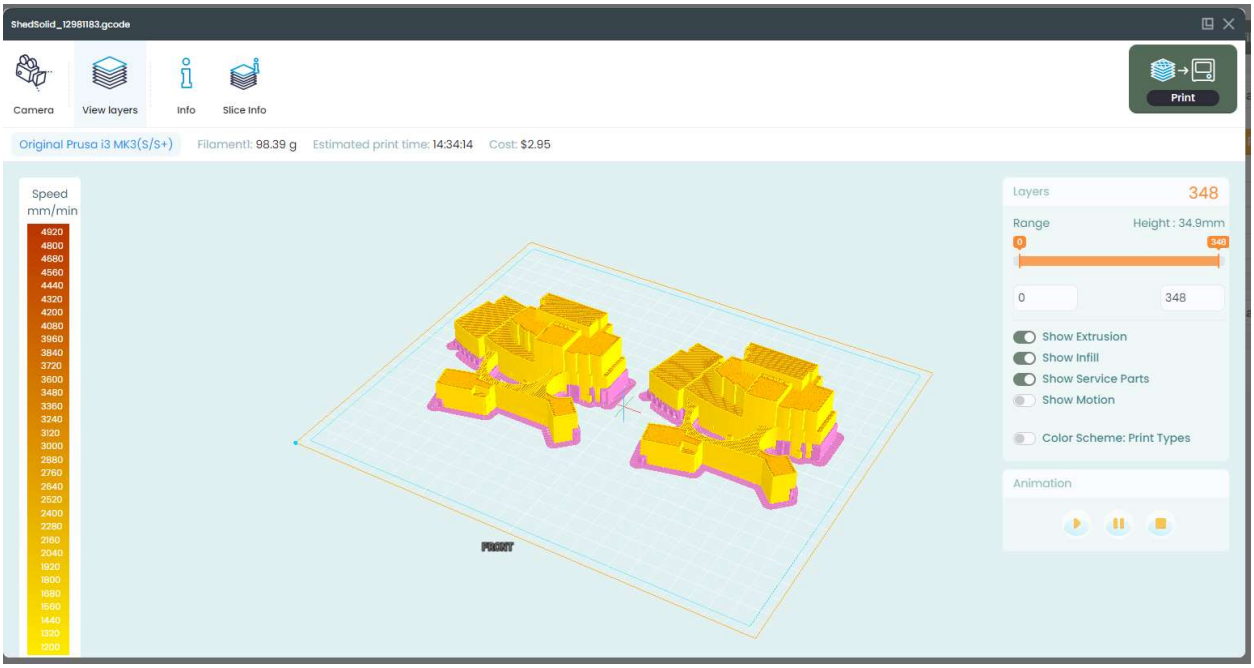


Once I have configured the settings how I want them to be, I can press either “Slice” at the bottom to slice the print and put it into the project, or “Slice & Toolpath Preview” to do the same as the other button, but also immediately open the G-code file to review. If you just slice, or want to go back and review G-code you generated in the past, you can find it in the project. The “Preview” button will let you analyze the G-code. Print can also be used to immediately start printing G-code from the project menu, but it is recommended to preview it first.



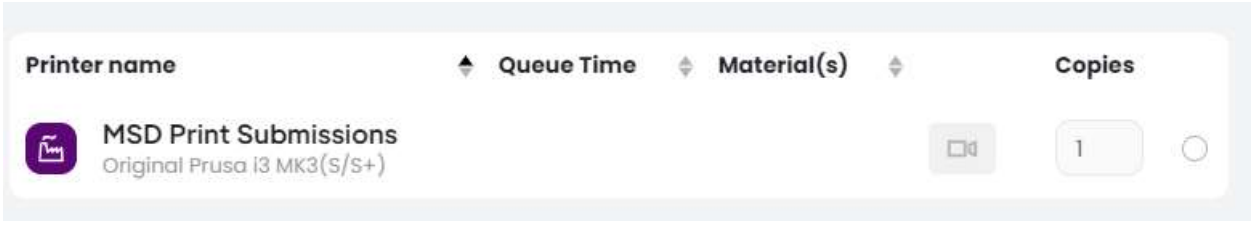
Note: It may take a few minutes for G-code to load for larger or more complex prints. You can always check back later in the “Projects” tab to see if your G-code has finished processing.

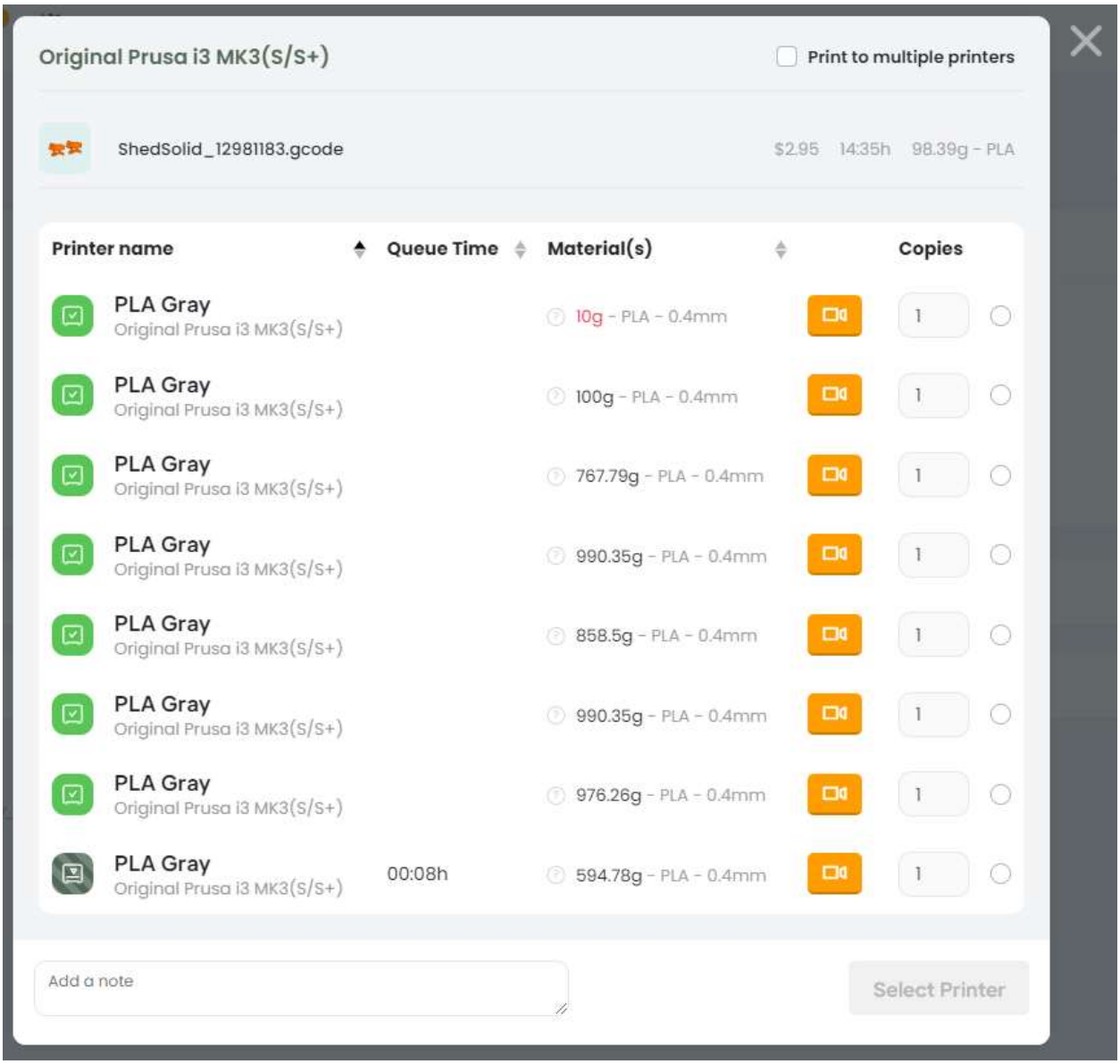
When we preview our G-code, we can see how much material it will use, how long it is estimated to take to print, and how much it will cost to print. The print will be color-coded, in the example below the support material is pink and the actual print is yellow. I can use the menus on the right to toggle visibility of certain parts of the G-code, and to run through an animation of the print.



If I see any issues in the G-code, or I want to change something, I can choose to slice the G-code from the "Projects" tab again. If I want to print this G-code, I can press the "Print" button in the top right. Doing so will bring up a list of all the 3D printers you have access to.

NOTE: Students in certain classes, clubs, and departments may see printers with a purple icon (example below). These are not normal printers, and printing to them will not work in the same way as the other printers you have access to. These are used either for special/complex printers that may require a machine operator to review before printing, or might be used for special classes or events. You can ignore these unless you are instructed specifically to use them.





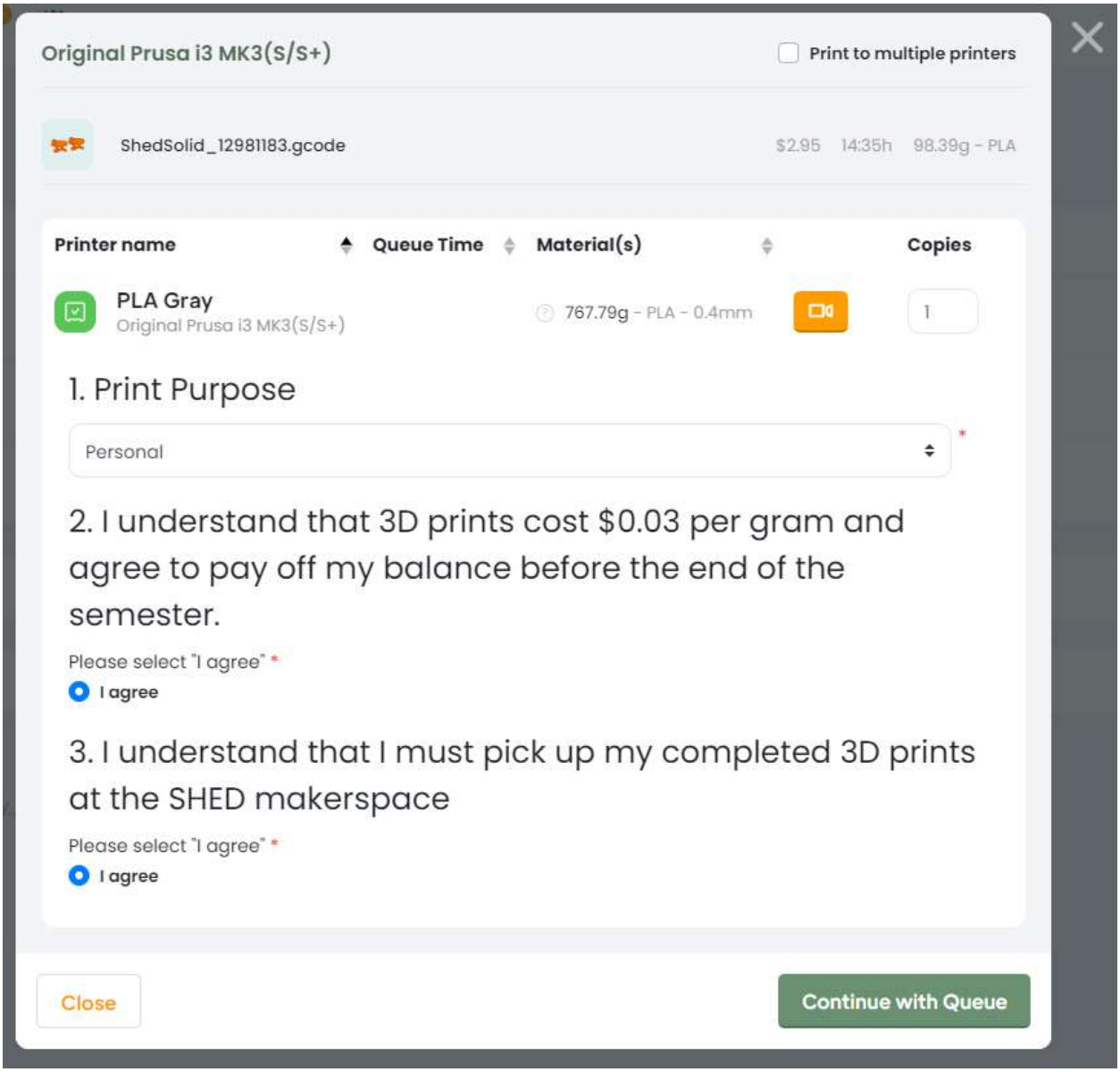
The printer name shows what material and color is currently loaded. In the example above, all printers I have access to have PLA Gray loaded. I can also see how many grams of PLA are on each machine, and in the top right is a reminder of how much filament my print takes. I should select a printer that has more filament remaining than my print will take to complete.

A green icon next to the printer means it is idle and ready to print. If you see a different icon, hovering over it will say what it is. For instance, the last printer in the list is currently printing.

In 3DPrinterOS, printers each have a queue. The queue allows you to send a print job to a printer that is already running, and your print will wait in line until it is your turn. This allows you to ensure you get a printer of the right color of filament, or lets you submit a print job even when all printers are running. You should try to choose the printer with the lowest queue time listed that still meets your needs, to ensure your print finishes as quickly as possible.

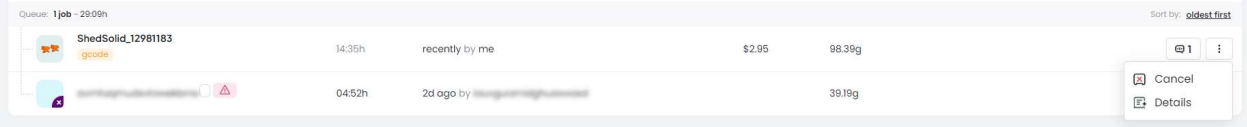
If you want to, you can print multiple copies of a print. At the time of writing, standard users are restricted to having 2 prints either in queue or printing at any given time. Since I am currently running a print, I would only be able to start 1 more. If I want to start multiple copies of the G-code, I can either enter a number of copies after selecting a printer, or hit “Print to Multiple Printers” at the top to choose multiple machines to print to.

Optionally, you can add a note to the print for the maker mentor to read. This can also be useful to keep notes for yourself about the print. There is no guarantee that a member of staff will see this note before the parts are printed, so do not put critical messages there. Please speak to a maker mentor directly before printing if you have a critical request.



When you select a printer, two buttons will appear at the bottom. For standard users, the only option available will be “Queue”. Pressing this button will bring up a few questions about your print you must fill out, and then you can press “Continue with Queue” at the bottom of the page.

Once your print is added to the queue, you can track it in the “Printers” tab. If your print has not yet started, you can cancel it by pressing the 3 dots next to the print’s name in the “Printers” tab, and then hitting “Cancel”.



You will receive automated email notification when your print is queued, when it starts printing, and when it finishes printing. You will also get a timelapse video of your print! If there are any issues while printing, you will receive an email detailing those as well.

If you have any questions or run into any issues, please consult a maker mentor at the SHED makerspace. You can also ask questions on our Slack channel; rit-makerspace.slack.com.