3DPrinterOS Guide

UNLV School of Architecture
Spring 2025

Introduction

3DPrinterOS is a cloud-based 3D printing management platform for managing files, machines, and users. It allows for remote control and monitoring, 3D model preparation, cloud slicing, user management, and 3D printer fleet management.

Contents

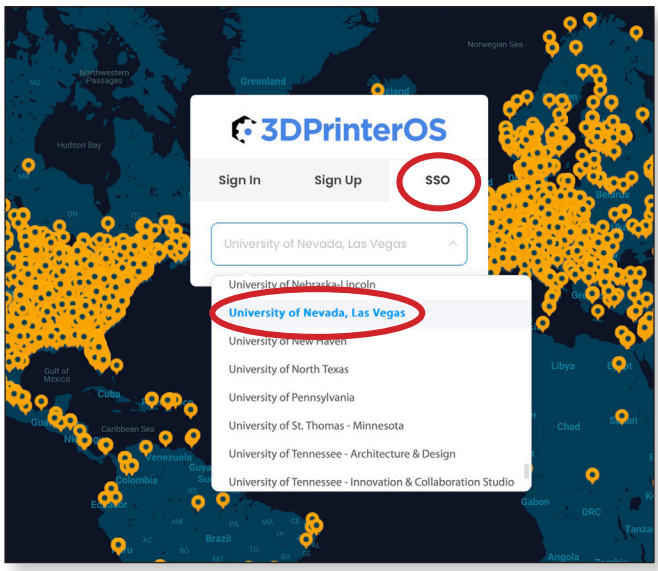
Section	*Title*	**Page #**
Standard File Preparation		
Part 1:	File Preparation	3
Part 2:	Bambu Lab Cloud Slicer	7
Part 3:	Flashforge Cloud Slicer	11
Part 4:	Print Queue	15
Advanced File Preparation		
Part 5:	Bambu Lab G-code	18

Part 1 File Preparation

1. Launch 3DPrinterOS

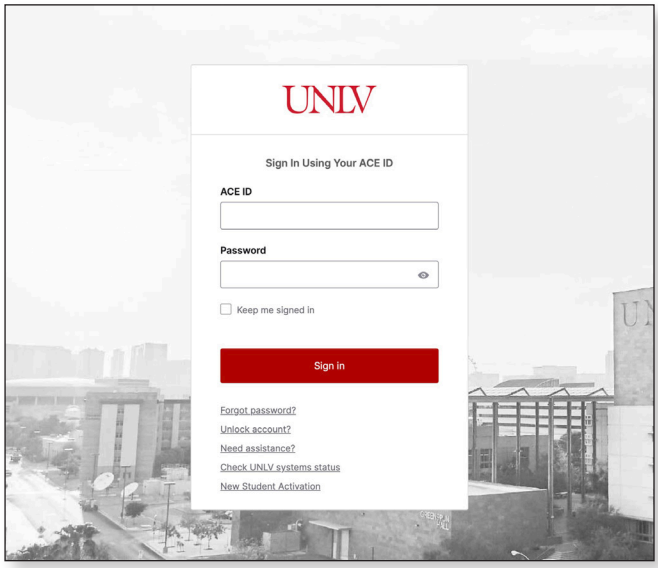
*Go to **https://cloud.3dprinteros.com** & select **SSO.***

*In the drop down menu select **University of Nevada, Las Vegas**.*



2. Log in to ACE Account

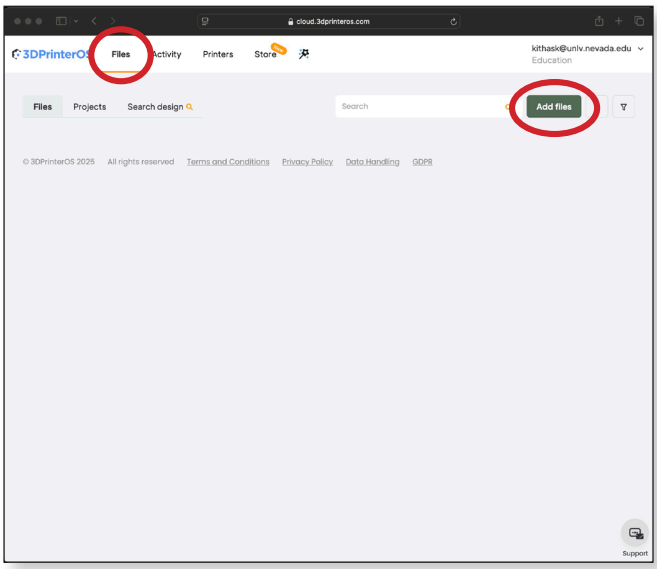
*Log in using your **ACE account** credentials.*

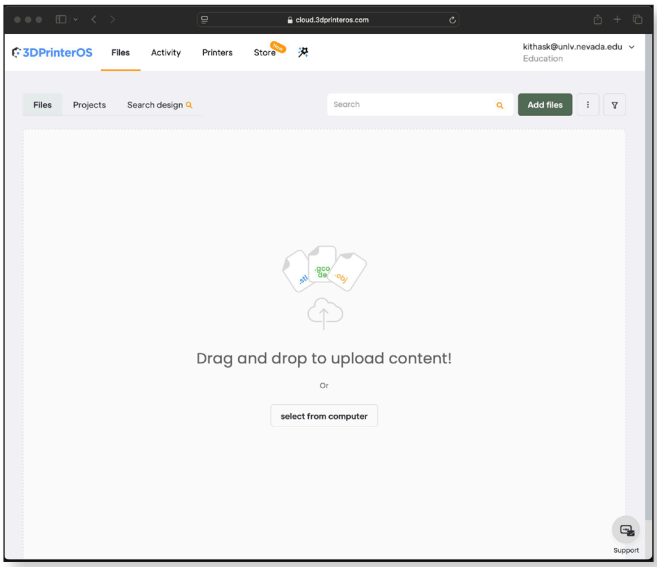


3. Upload Your 3D Model

*Under the **Files** tab, click the **Add Files** button in the upper-right corner.*

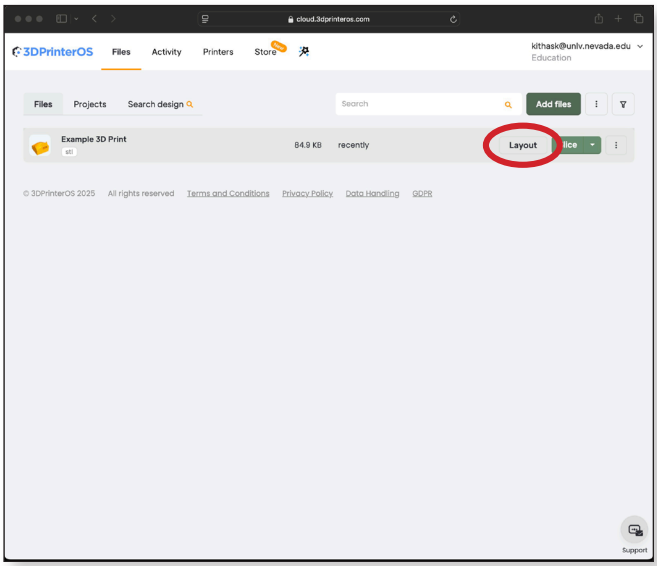
Upload your 3D model file (STL, OBJ, etc.).





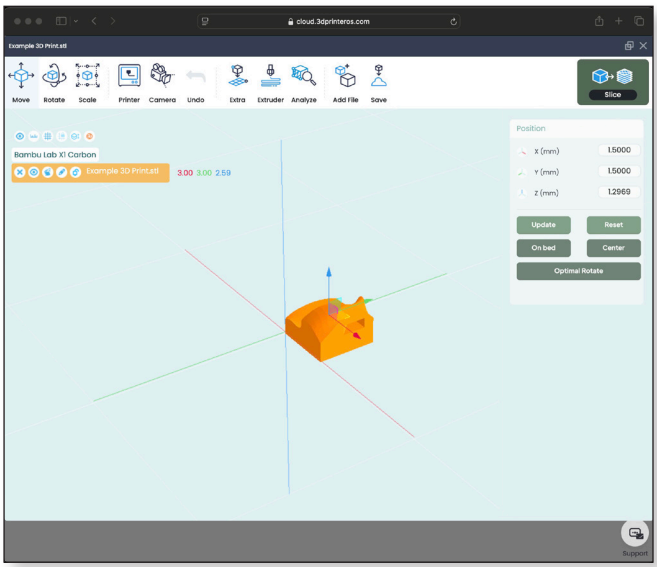
4. Position Your Model

*Once the file is uploaded, click the **Layout** button.*



5. Layout

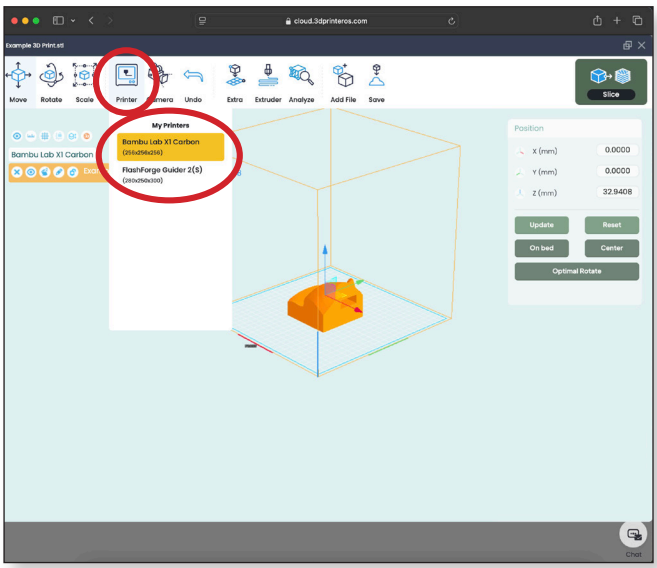
*The **Layout** screen provides options to ensure your part is correctly positioned.*



6. Select A Printer

*Click the **Printer** button to choose which printer you want to use.*

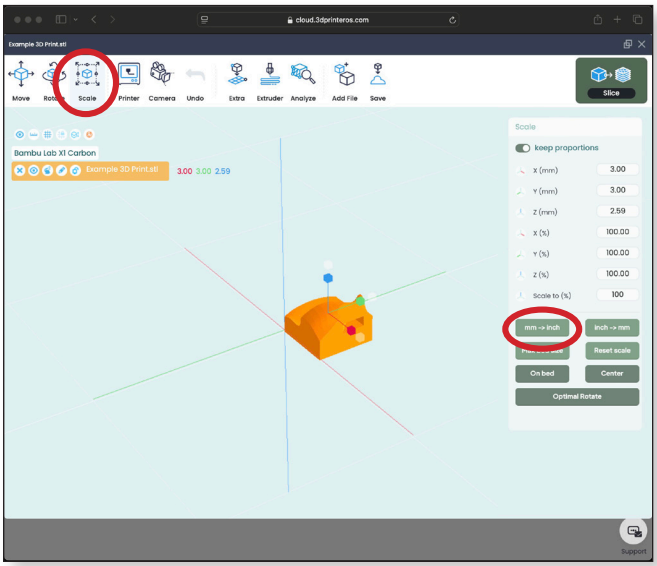
> ***Tip:** Before selecting a printer consider what filament type and color you would like to use.*
>
> *Refer to our website to see what filaments are available for each printer.*



7. Scale

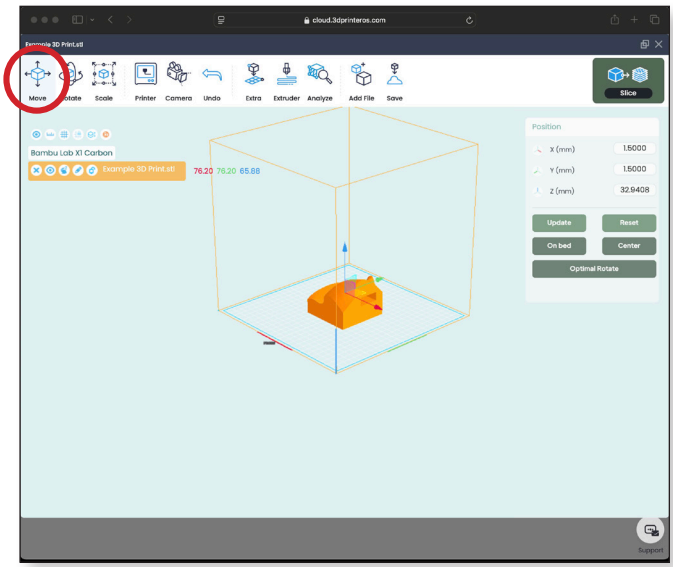
Ensure the model units are correct.

*If you have designed your part in inches, click the **mm -> Inch** button.*



8. Move

Check that the part is within the printer bounding box.



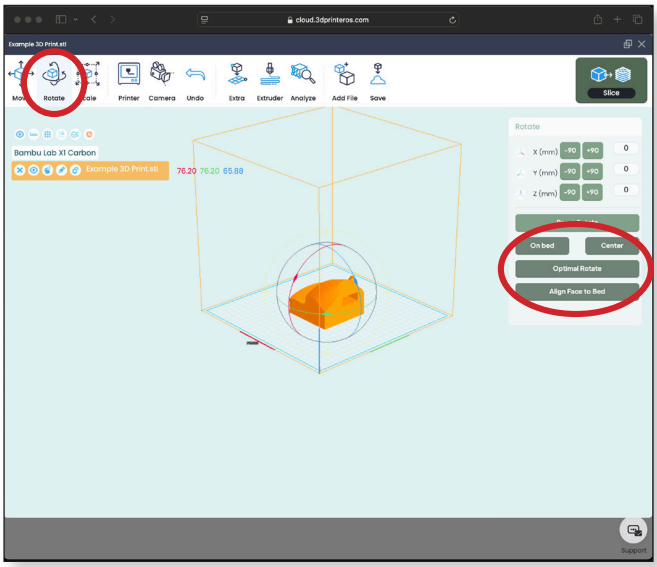
9. Rotate

Part orientation can impact print time and quality.

*Click the **On Bed** button to ensure the part lies on the print bed.*

*Click the **Optimal Rotate** button to verify optimal orientation.*

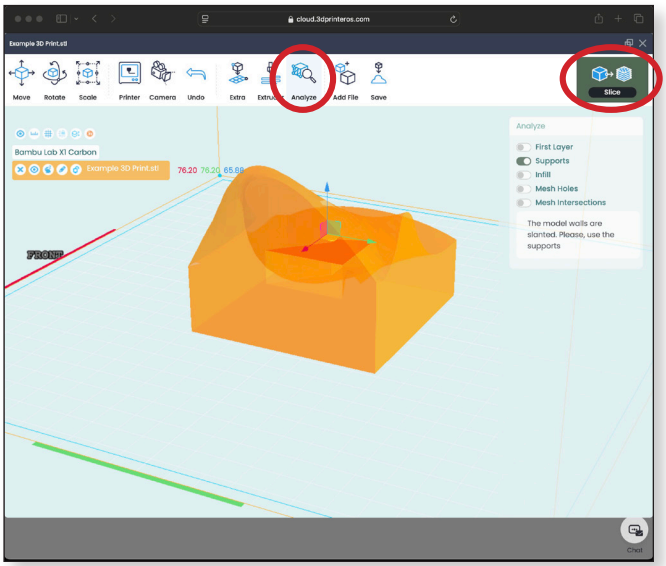
*Click the **Align Face to Bed** button to align a specific face to the print bed.*



10. Analyze Your Model

Check if your model needs supports or if repairs are required.

*Once satisfied with the layout, click the **Slice** button in the top-right corner.*



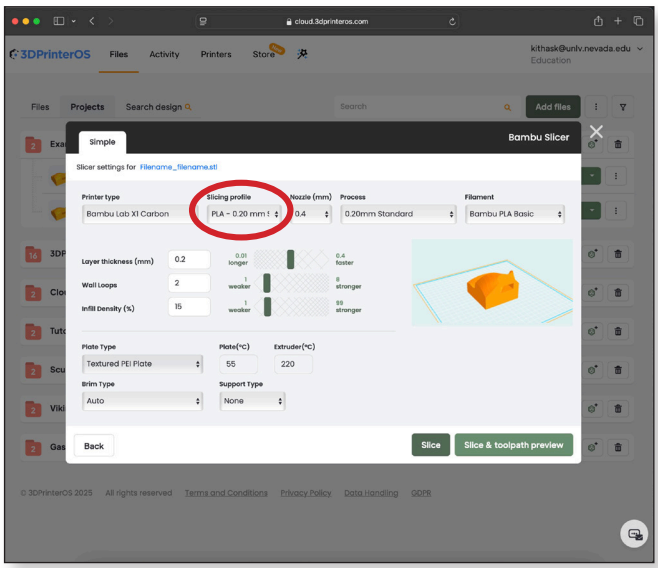
Part 2 Bambu Lab Slicer

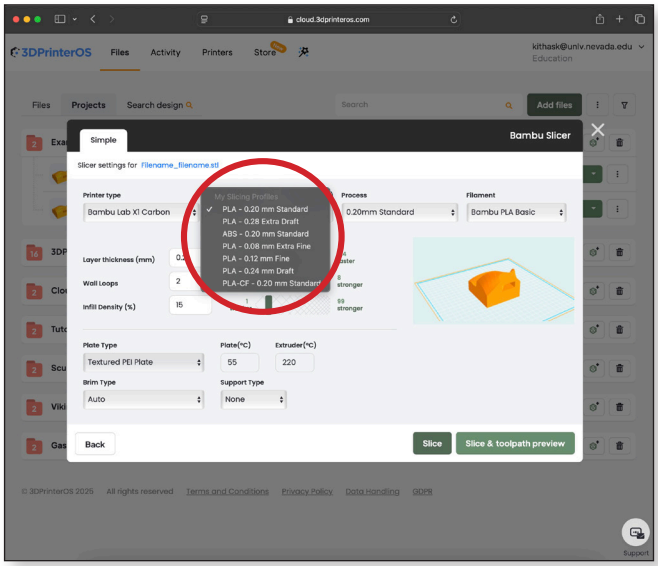
1. Slice Settings

*Select a **Slicing Profile.***

*The Slicing Profile will auto populate the following: **Nozzle (mm)**, **Process**, **Filament**, **Layer Thickness (mm)**, **Wall Loops**, and **Infill Density (%)**.*

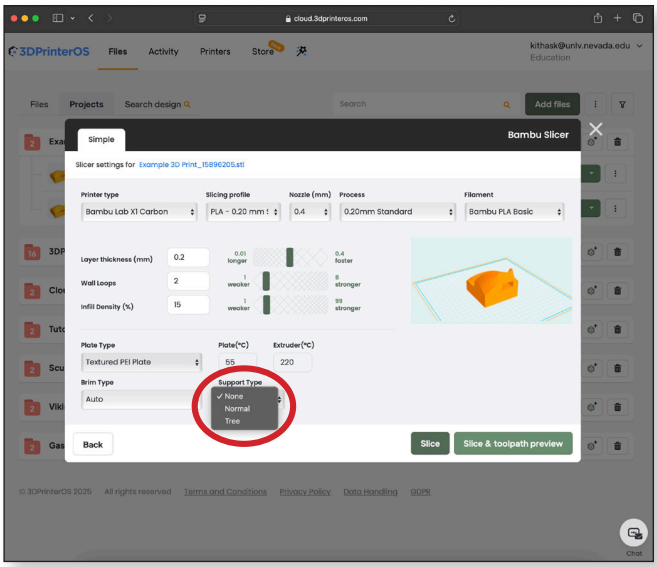
> ***Tip:** If you are supplying your own filament use the filament drop down to select the specific brand of filament you are supplying after selecting a slicing profile. If the brand is not listed, select **Generic**.*





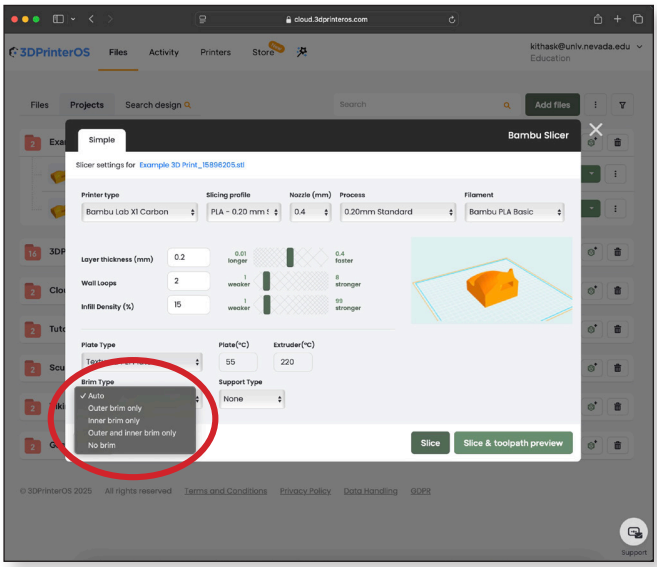
2. Supports

*If your model needs supports click the **Support Type** drop down menu and select **Normal** or **Tree Supports.***



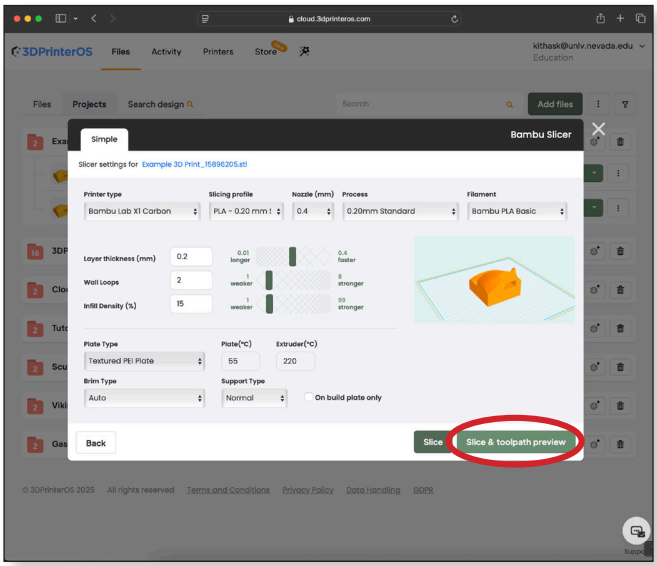
3. Brim

*To improve bed adhesion, especially for parts with small footprints, add an **Outer Brim**.*



4. Slice Model

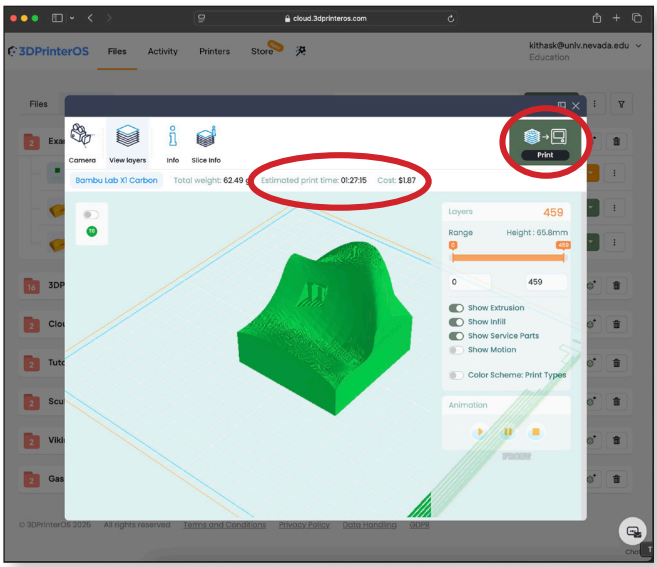
*Once satisfied with the slicing settings, click the **Slice & Toolpath Preview** button.*



5. Toolpath Preview

A preview of your print will appear, showing an estimated print time and filament cost.

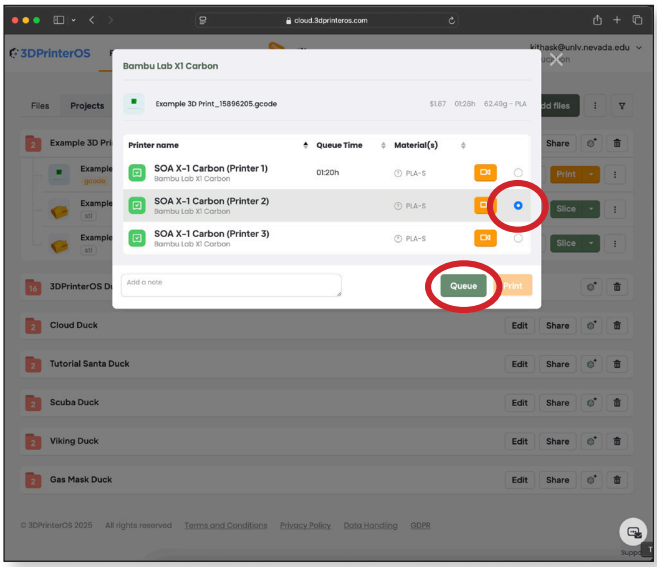
*If you're satisfied with the preview, click **Print** in the upper-right corner.*



6. Select Printer

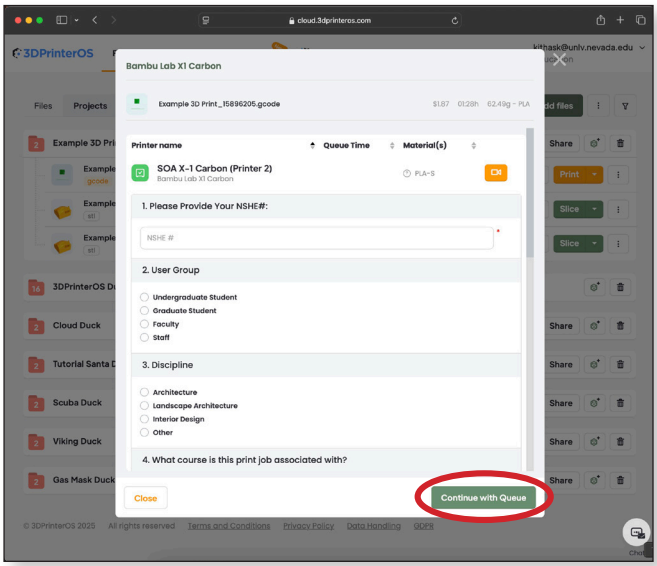
A list of available printers will appear. Select the printer with the lowest queue time.

*Click the **Queue** button in the bottom-right corner.*



7. Submittal Form

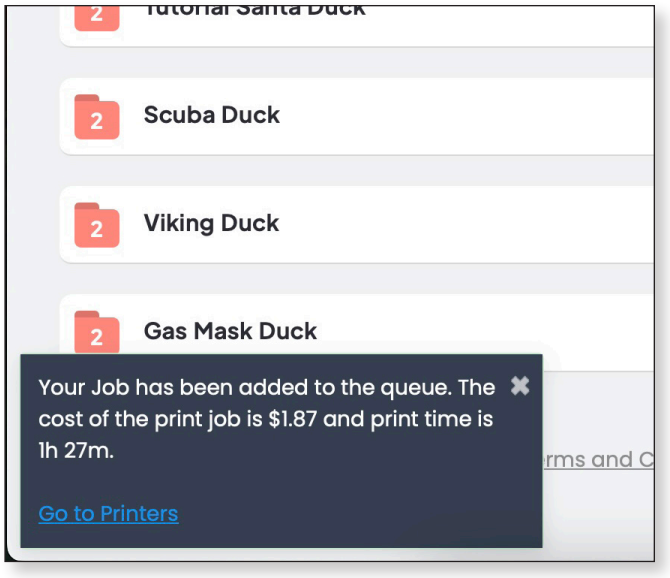
*Complete the 3D Print submittal form, then click the **Continue with Queue** button.*



8. Job Has Been Added to Queue

*You'll receive a message saying, "**Your Job has been added to the queue"**.*

Lab staff will review and release your print job to the printer.

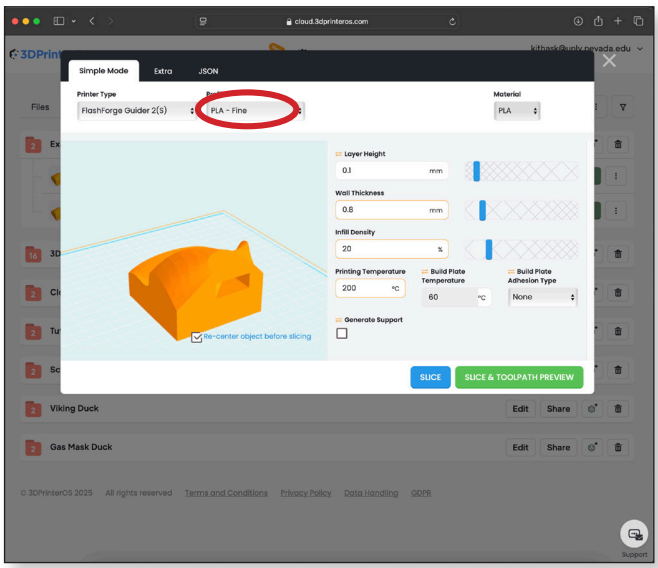


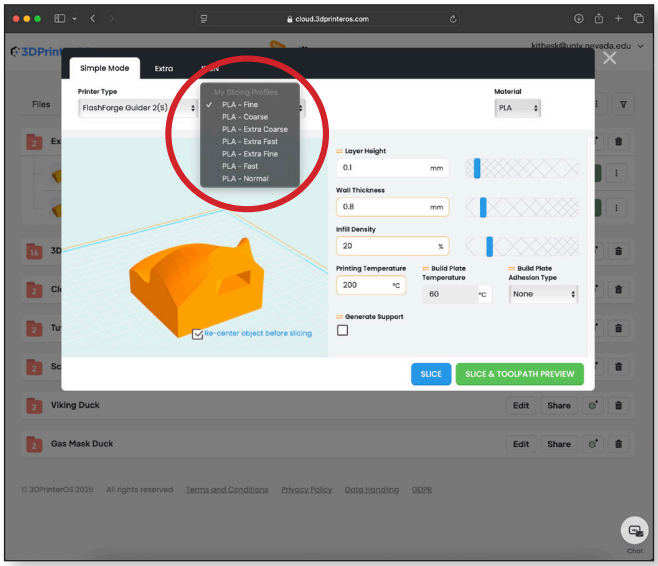
Part 3 Flashforge Slicer

1. Slice Settings

*Select a **Slicing Profile.***

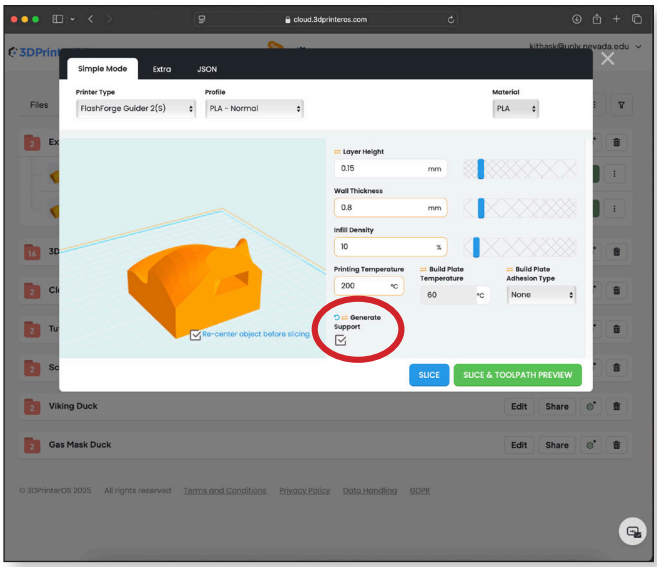
*The Slicing Profile will auto populate the following: **Material**, **Layer Height (mm)**, **Wall Thickness (mm)**, and **Infill Density (%)**.*





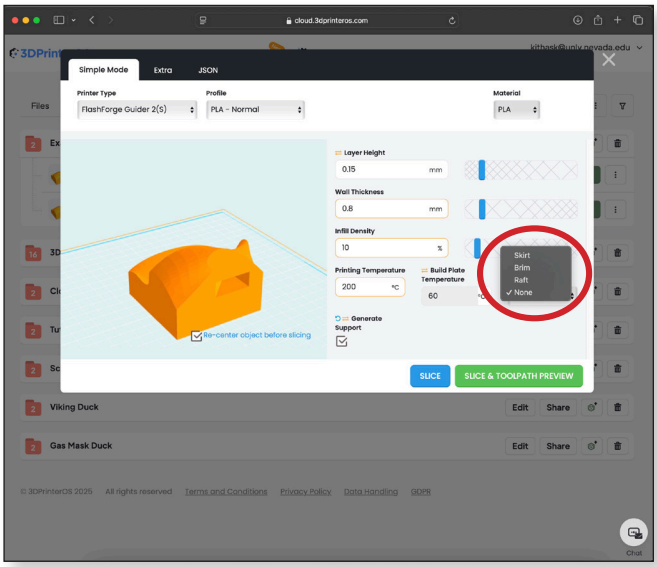
2. Supports

*If your model needs supports check **Generate Support.***



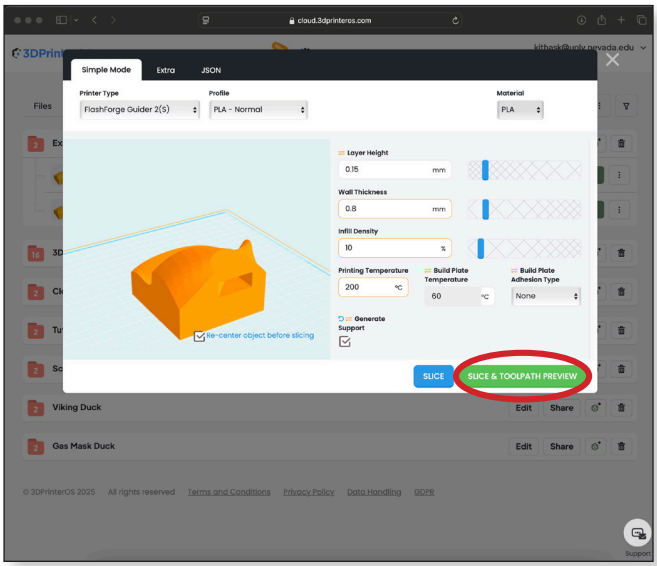
3. Brim

*To improve bed adhesion, especially for parts with small footprints, add a **Brim**.*



4. Slice Model

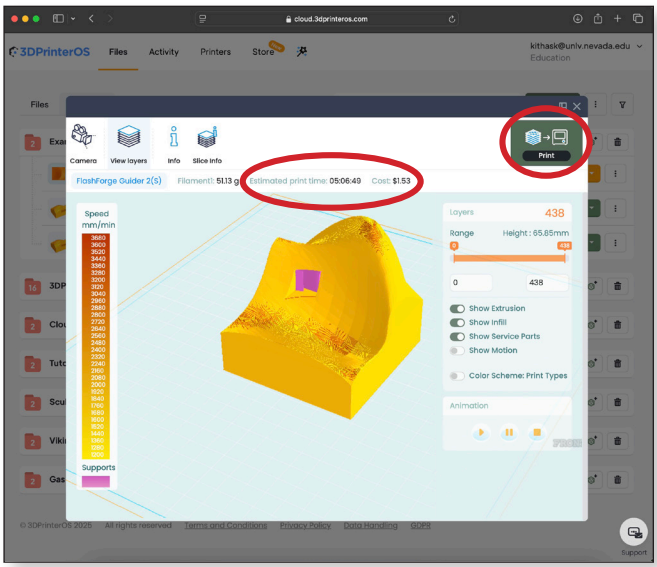
*Once satisfied with the slicing settings, click the **Slice & Toolpath Preview** button.*



5. Toolpath Preview

A preview of your print will appear, showing an estimated print time and filament cost.

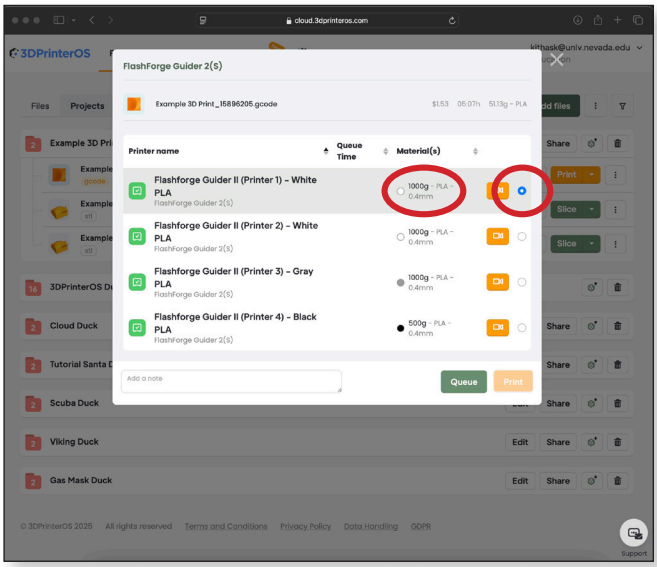
*If you're satisfied with the preview, click **Print** in the upper-right corner.*



6. Select Printer

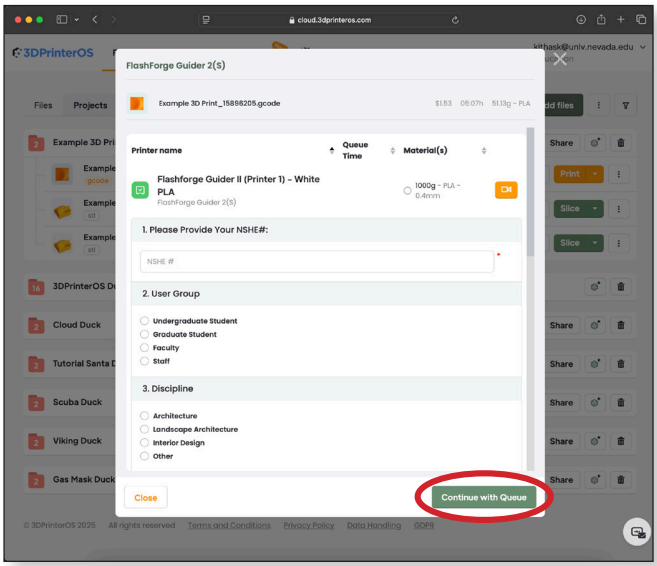
A list of available printers will appear. Select the printer with the color filament you want to use.

*Click the **Queue** button in the bottom-right corner.*



7. Submittal Form

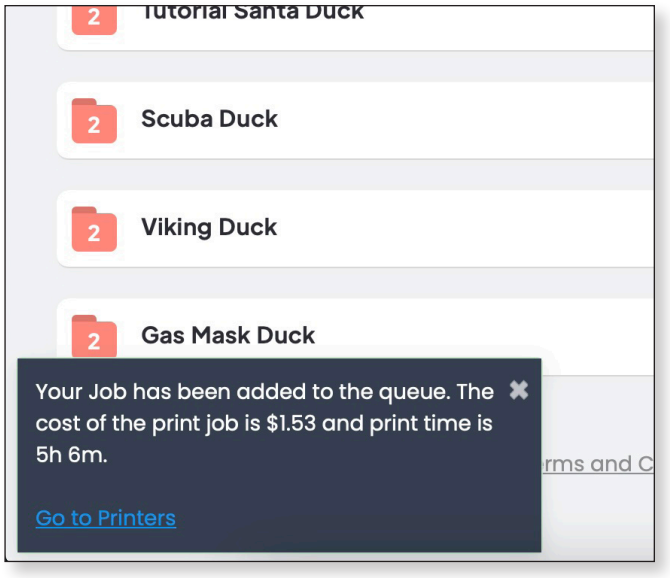
*Complete the 3D Print submittal form, then click the **Continue with Queue** button.*



8. Job Has Been Added to Queue

*You'll receive a message saying, "**Your Job has been added to the queue"**.*

Lab staff will review and release your print job to the printer.

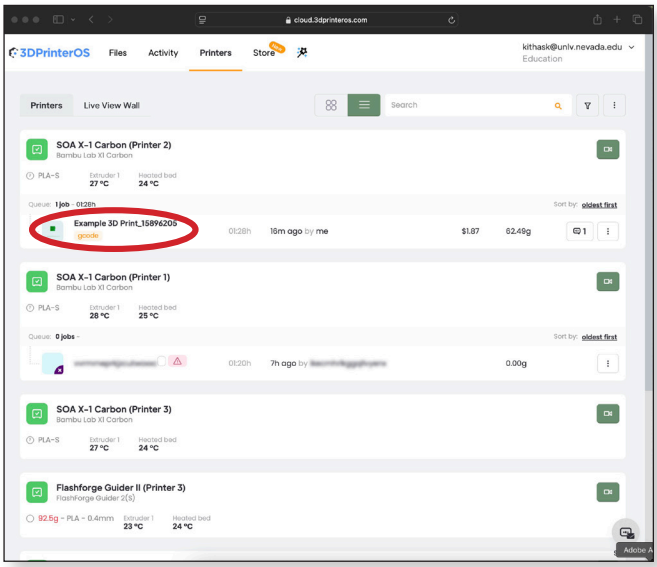


Part 4 Print Queue

1. Print Queue

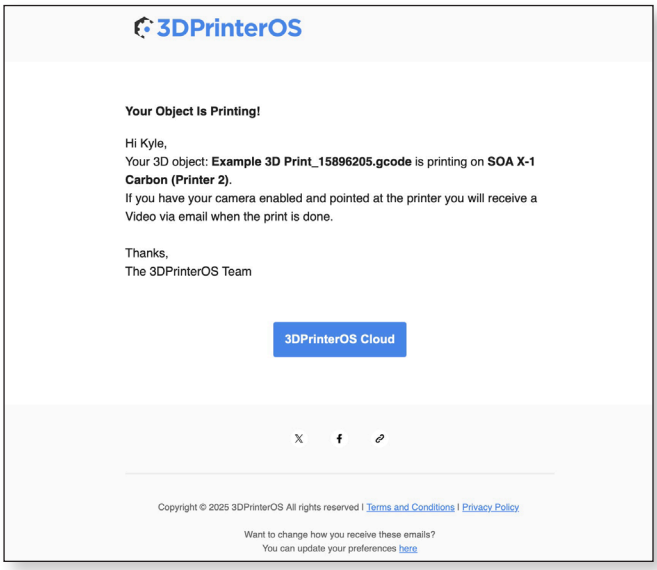
*Under the **Printers** tab, the print queues are visible for each printer.*

***Tip:** To cancel a print job in the queue before it has been sent to the printer click the button with the three dots on the right side of the print job and click **Cancel** in the drop down menu. Once a job is sent to the printer it can not be canceled.*



2. Print Sent to Printer

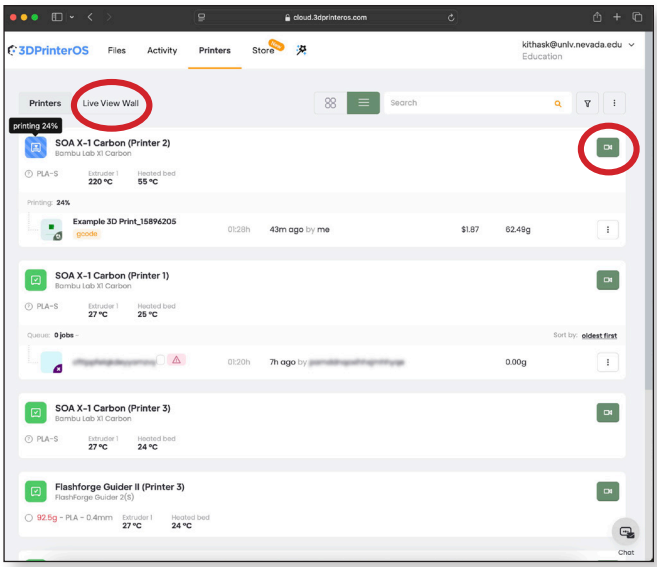
You will receive an email notification when lab staff has reviewed your job and sends it to the printer.



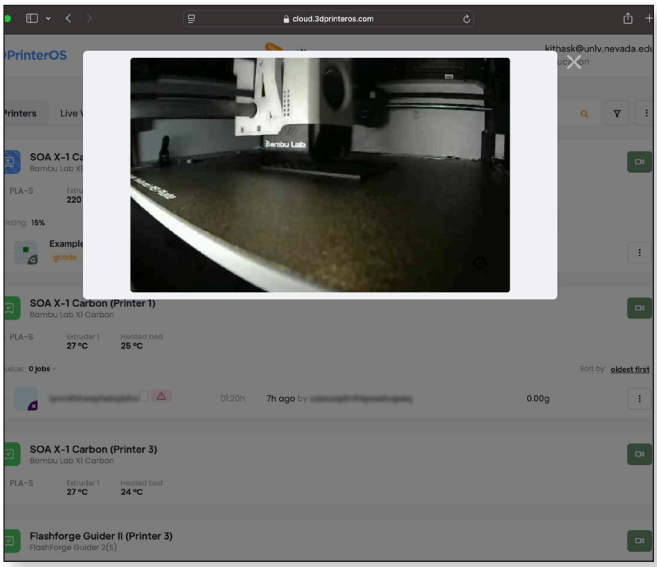
3. Monitor Your Print

*You can get a live video feed of the printer and monitor your print job by clicking the green **Camera** button next to the printer.*

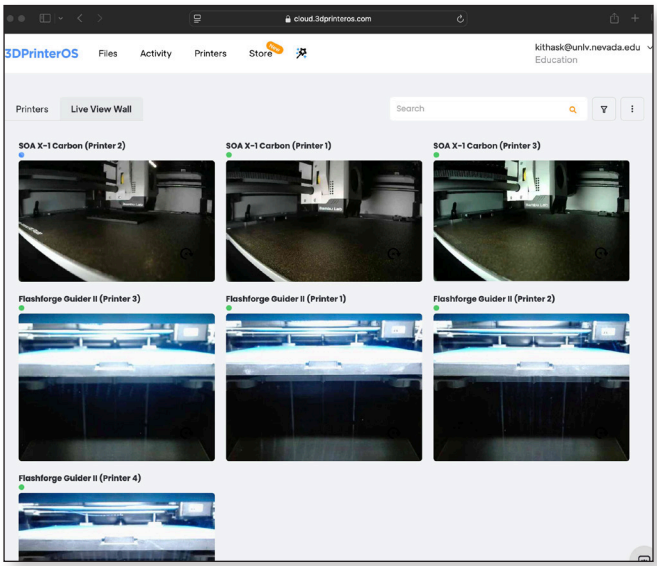
*Alternatively, you can get live video feeds of every printer by clicking the **Live Video Wall** tab.*



4. Live Camera Feed



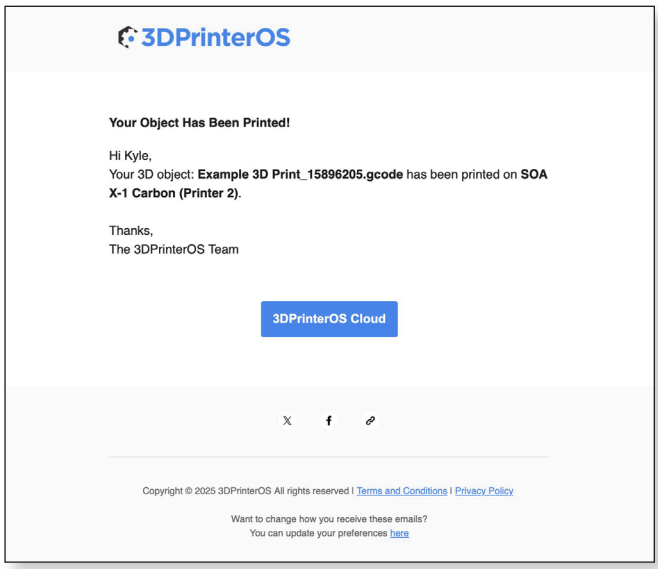
5. Live Video Wall



6. Print Job Complete

You will receive an email notification when your print job is complete.

Visit the lab during operational hours to pick up your print and pay for the filament used.



7. Time Lapse Video

Part 5 Bambu Lab G-code

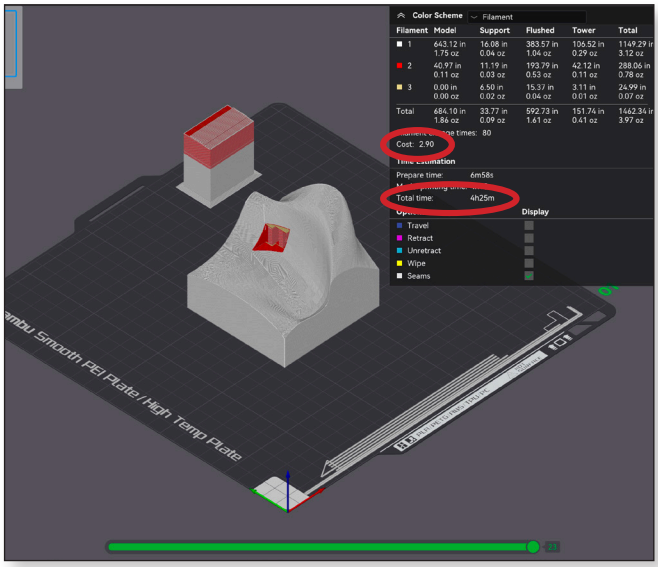
1. Export G-code From Bambu Studio

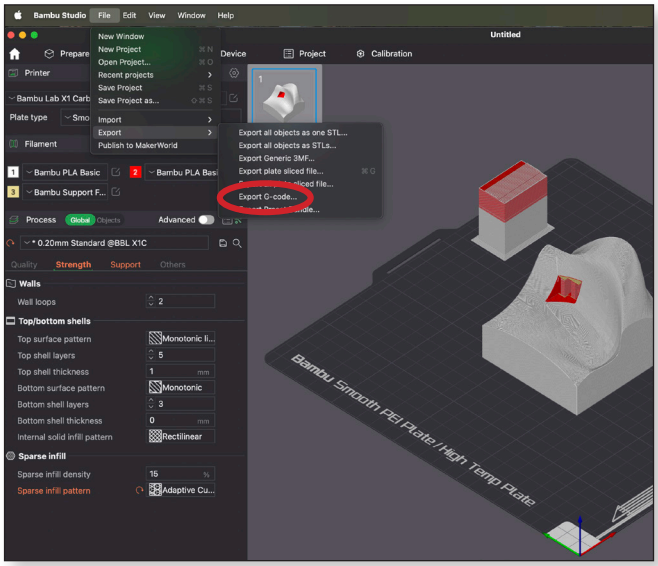
This 3D model requires slicing in Bambu Studio because of the following:

1. ***Multicolor Printing:*** *White + Red PLA*
2. ***Advanced Support Material:*** *Bambu Support for PLA as support/raft interface*
3. ***Advanced Internal Infill Patterns:*** *Adaptive Cubic*

*Typically we recommend using the **textured PEI plate** because it provides optimal adhesion. However, if there is a need for a smooth bottom, the **smooth PEI plate** can be used. **The smooth PEI plate is opted for in this example.***

*Navigate to **Export** in the **File** menu and click **Export G-code...***





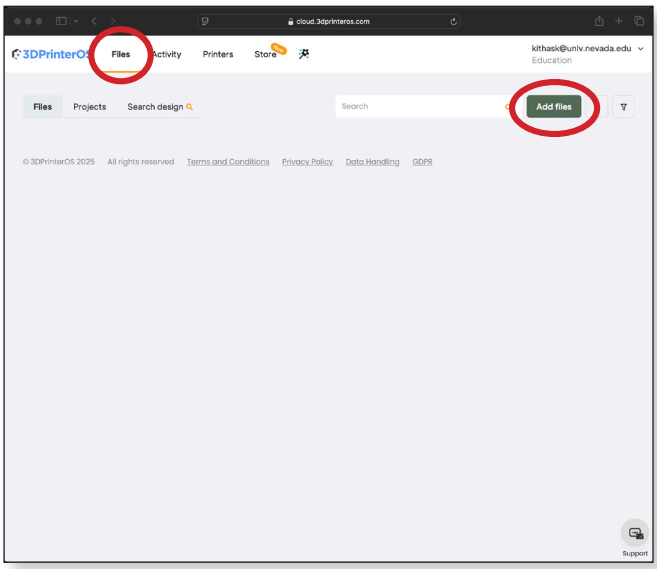
Tip: *When preparing multicolor prints, ensure each part of the model is assigned the correct color.*

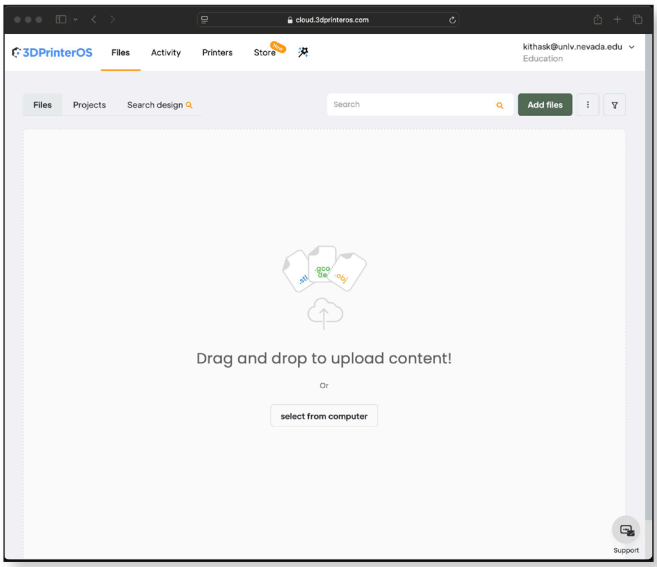
Tip: *To improve bed adhesion, especially for parts with small footprints, add brims.*

2. Upload Your G-code

*Under the **Files** tab, click the **Add Files** button in the upper-right corner.*

Upload your G-code file.

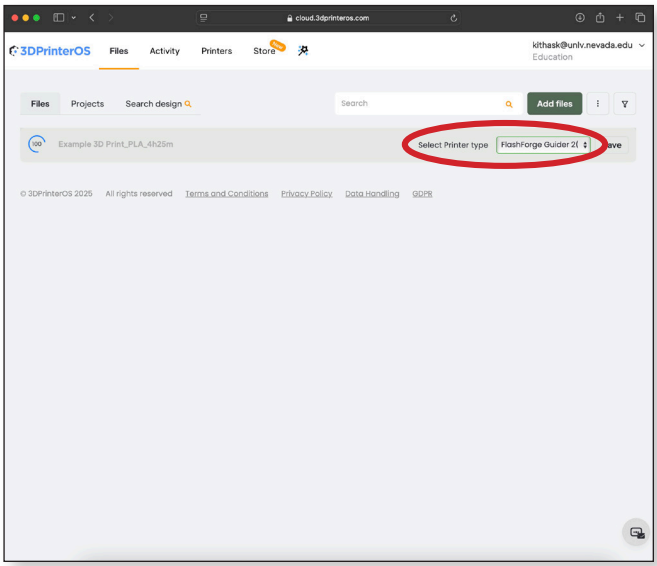


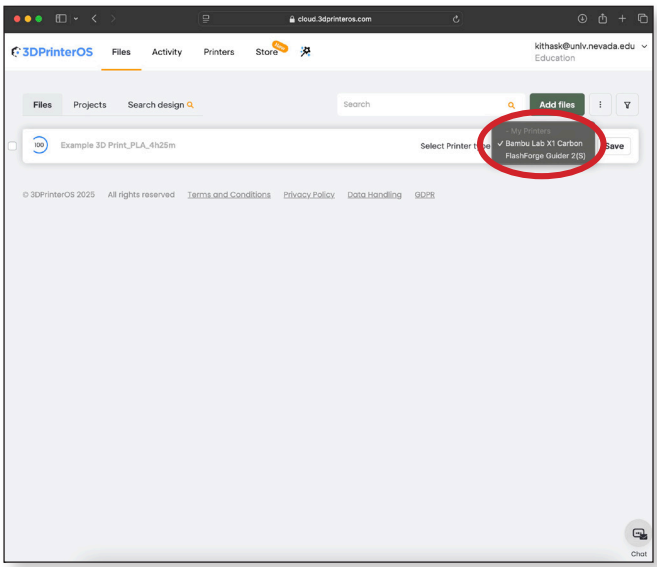


3. Select Printer Type

*Once the file is uploaded, select **Bambu Lab X-1 Carbon** from the select printer type drop down menu.*

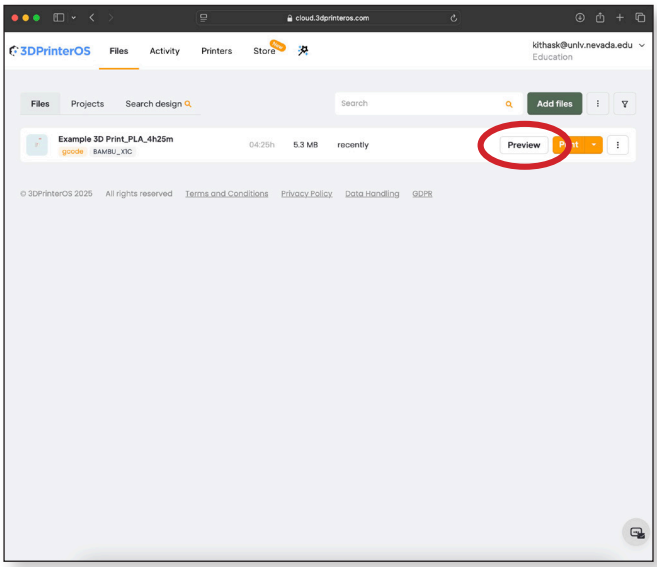
*Click **Save**.*





4. Preview

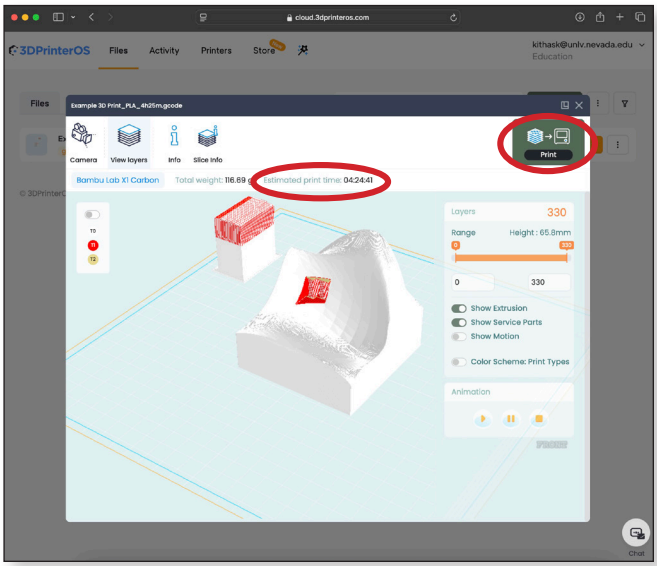
*Once the file is uploaded, click the **Preview** button to ensure your file imported correctly.*



5. Toolpath Preview

A preview of your print will appear, showing an estimated print time.

*If you're satisfied with the preview, click **Print** in the upper-right corner.*

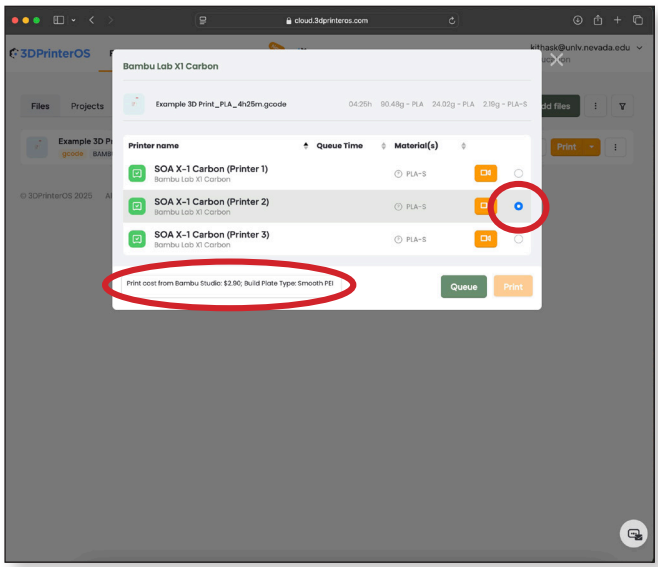


6. Select Printer

A list of available printers will appear. Select the printer with the lowest queue time.

***Important:** In the notes section input the **cost** from Bambu Studio and the **build plate type**.*

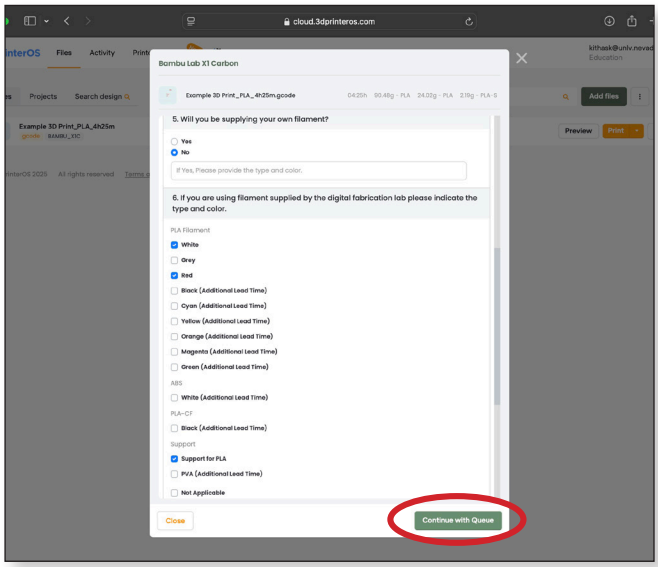
*Click the **Queue** button in the bottom-right corner.*



7. Submittal Form

*Complete the 3D Print submittal form, then click the **Continue with Queue** button.*

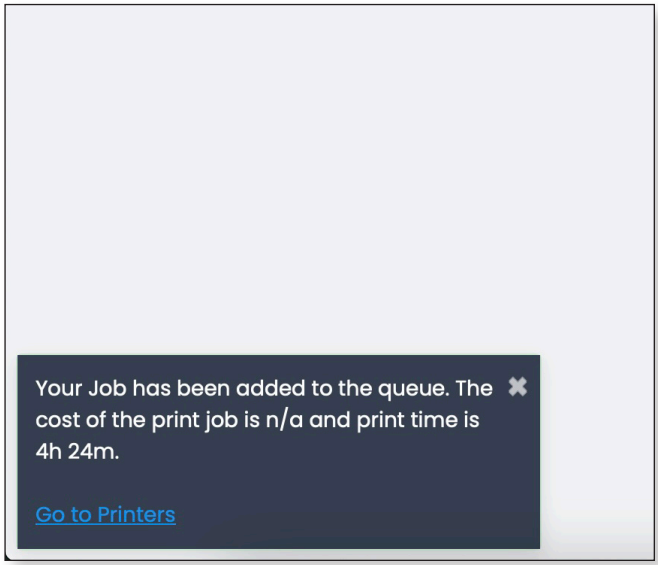
***Important:** When printing with multiple filaments, **check** all that apply. A **maximum of 4 filaments** can be used for a print.*



8. Job Has Been Added to Queue

*You'll receive a message saying, **"Your Job has been added to the queue"**.*

Lab staff will review and release your print job to the printer.



***Tip:** The cost of the print job will display as **n/a**. This is expected because cost information is not transferred with the G-code file.*