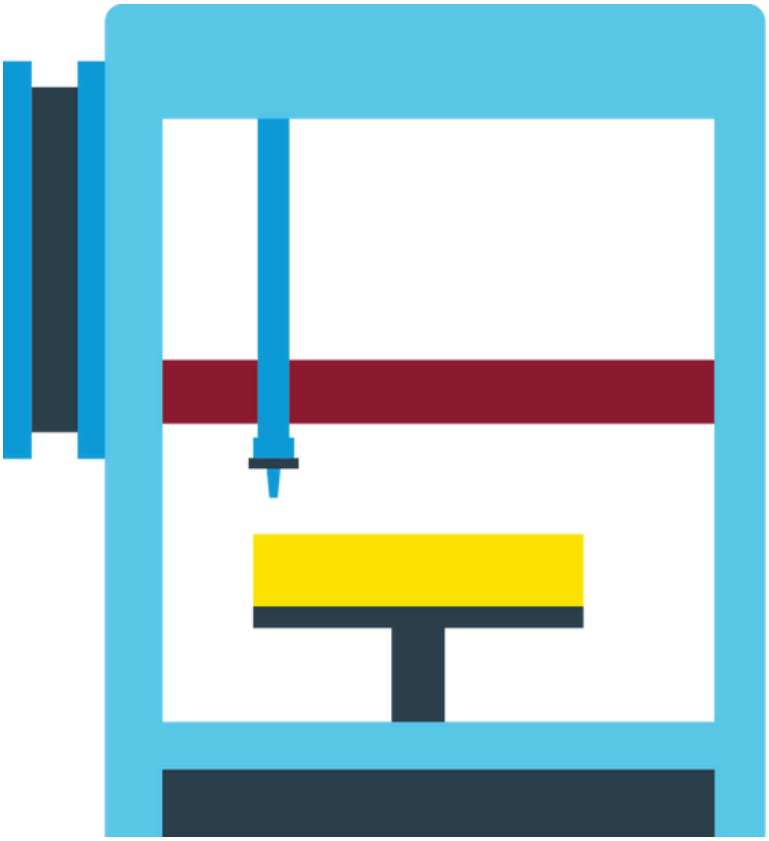
MacEwan UNIVERSITY | LIBRARY

MAKERSPACE

3D PRINTING GUIDE

FDM Machines



CC BY-NC 4.0
Creative Commons Attribution-NonCommercial 4.0 International

OVERVIEW

What is 3D printing?

The process of building a physical 3D object one thin layer at a time from a digital file. We use Fused Deposit Modelling (FDM) printers which push filament through a heated nozzle and deposit it layer by layer.

This guide includes:

Machine Specifications

Ultimaker 2 extended

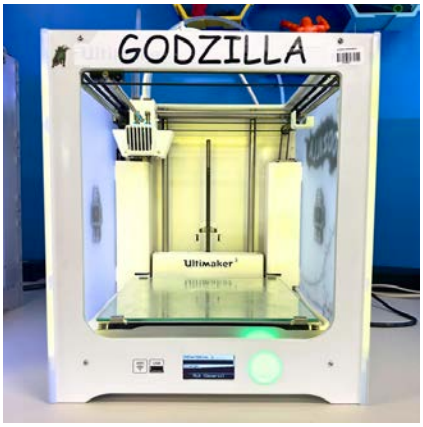
Official Website

Number of Printers: 1
Build Volume: 8.77" x 8.77" x 12"
Our Machine Names: Cthulu

Ultimaker S3

Official Website

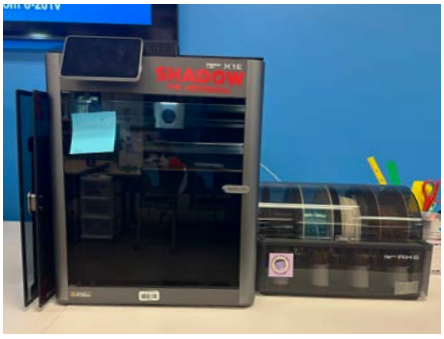
Number of Printers: 2
Build Volume: 9" x 7.4" x 7.9"
Our Machine Names: Audrey II & Dracula



Ultimaker 3

Official Website

Number of Printers: 1
Build Volume: 9" x 7.4" x 7.9"
Our Machine Names: Godzilla & Slimer



Bambu Lab X1e

Official Website

Number of Printers: 1
Build Volume: 10" x 10" x 10"
Our Machine Names: Shadow The Hedgehog

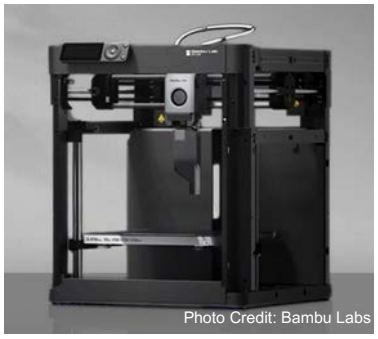
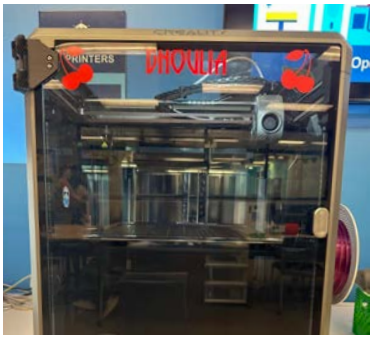


Photo Credit: Bambu Labs

Bambu Lab P1P

Official Website

Number of Printers: 2
Build Volume: 10" x 10" x 10"
Our Machine Names:

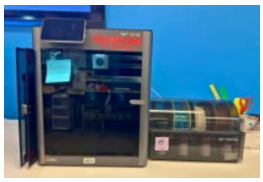
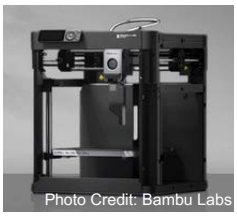


Creality K1 Max

Official Website

Number of Printers: 2
Build Volume: 12" x 12" x 12"
Our Machine Names: Ghoulia & The Silence

Printer Use Cases

Printer Type		Photo	Speed	Size	Colours
Bambu Lab X1e (Shadow the Hedgehog)			Fast	Medium to large size prints	Up to 4 colours
Bambu Lab P1P		 Photo Credit: Bambu Labs	Fast	Medium to large size prints	1 colour
Creality K1 Max (Ghoulia & The Silence)			Fast	Medium to large size prints	1 colour
Ultimaker	2 Extended (Cthulu)		Slower	Small to medium sized prints	1 colour
	S3 (Audrey II & Dracula)				Up to 2 colours
	3 (Godzilla & Slimer)				

SAFE USE

Safety

Do not touch the nozzle/hot end of the 3D printers during use or immediately after use.

Nozzles reach temperatures over 200°C (392°F) — touching them can cause severe burns.

Do not put your hand inside 3D printer while a job is going.

Motors, belts, and gears can pinch or trap fingers during operation.

Be careful when using hand tools.

Removing supports may require sharp tools (e.g., pliers, snips, utility knives) which can cause cuts or injury.

Be considerate of recycling limitations for 3D prints.

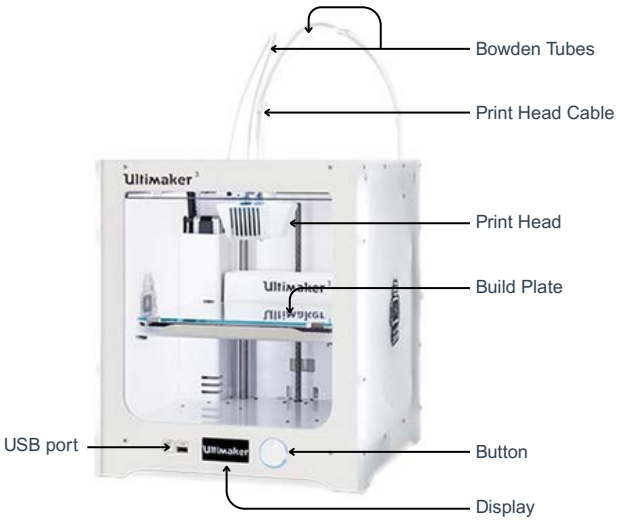
Most 3D printing plastic is not easily recyclable through standard programs.

Terminology

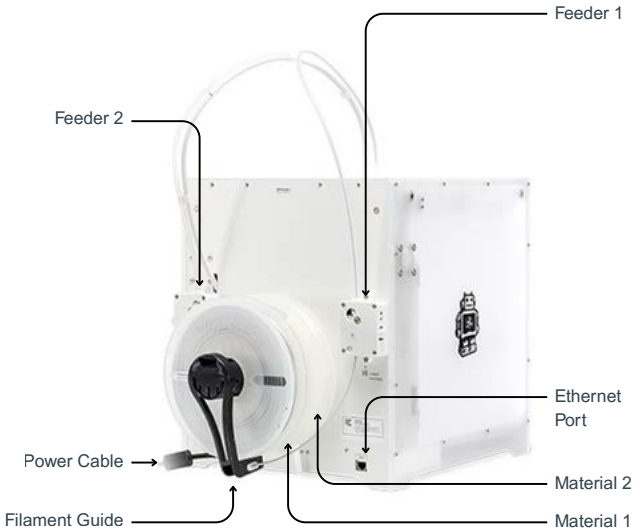
Term	Definition
Filament	Thermoplastic material used for FDM printing (e.g., PLA, ABS, PETG, TPU).
Polylactic Acid (PLA)	A beginner-friendly, biodegradable filament that's easy to print with.
Slicing	The method of taking a 3D model file and converting it to a file format that the 3D printers can understand.
.stl	A file format used to store 3D models. Best format for printing.
.gcode	A file format that tells a 3D printer how to move and print a model.
Build Plate / Print Bed	The flat surface where the print is built.
Extruder	The component that pushes the filament into the hot end for melting and deposition.
Hot End & Nozzle	The heated part of the extruder that melts the filament and the tip that extrudes it.
Support	Removable printed elements assisting overhangs or creating bridges.
Overhangs	Sections of a model that jut out beyond the layers beneath them.
Bridges	Horizontal stretches of material that span between two raised points.

Machine Diagrams

Ultimaker 3



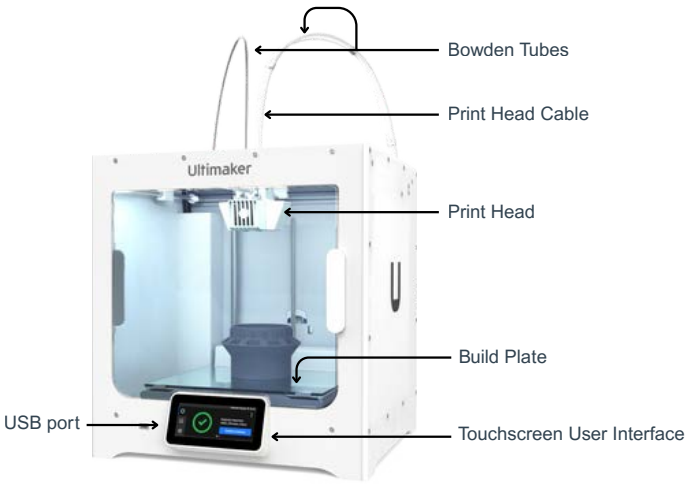
Front



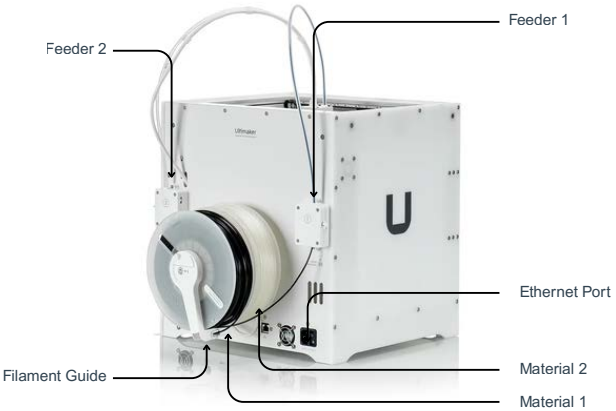
Back

Photo Credit: Ultimaker
https://www.dynamism.com/ultimaker/ultimaker-3.html

Ultimaker S3



Front

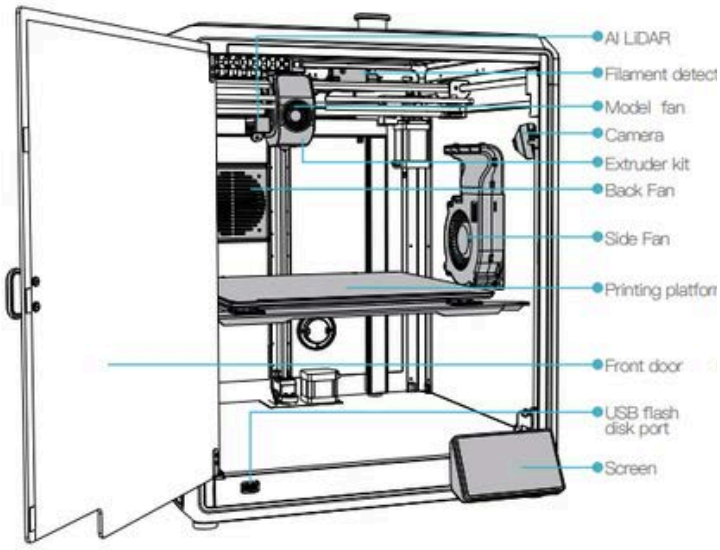


Back

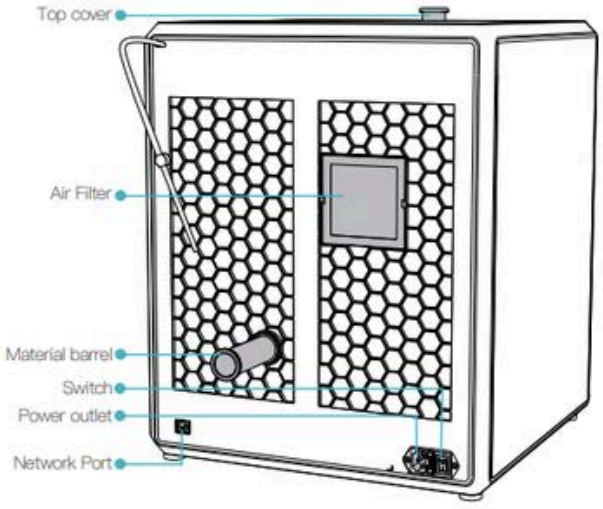
Photo Credit: Ultimaker
https://ultimaker.com/3d-printers/s-series/ultimaker-s3/

Machine Diagrams

Creality K1 Max



Front



Back

Photo Credit: Creality Wiki
https://wiki.creality.com/en/k1-flagship-series/k1-max/quick-start-guide/users-manual

Machine Diagrams

Bambu Labs X1E & P1P (X1 Carbon is shown but labeling is the same)

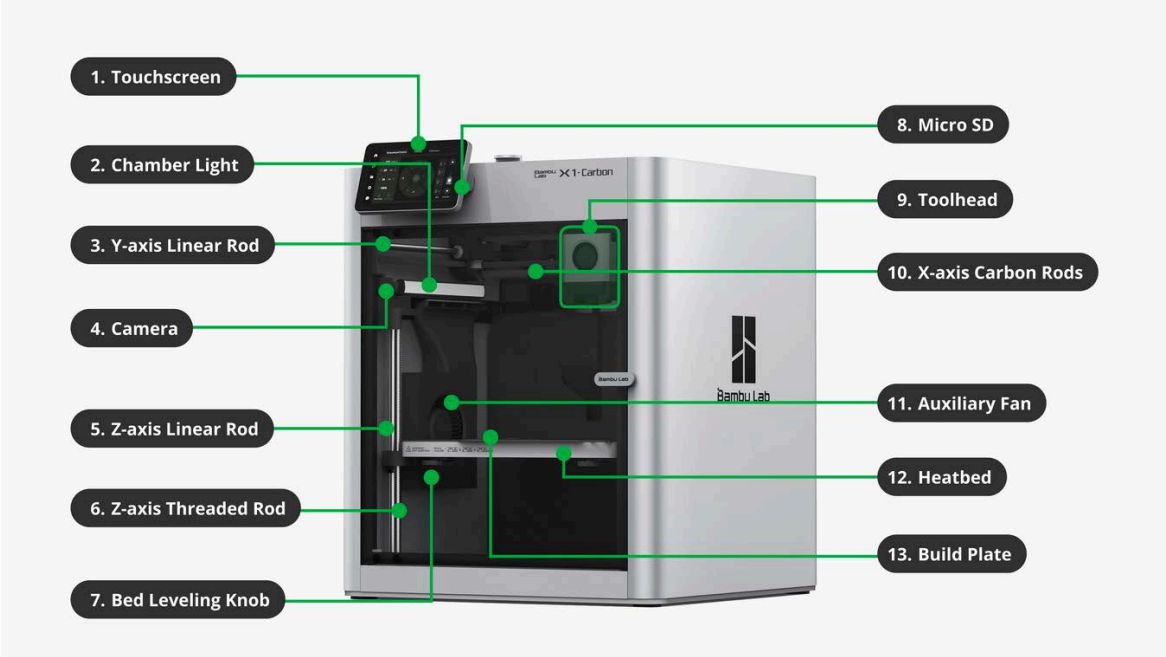


Photo Credit: Bambu Labs Wi
https://wiki.bambulab.com/en/x1ki

AMS (Automatic Material System)

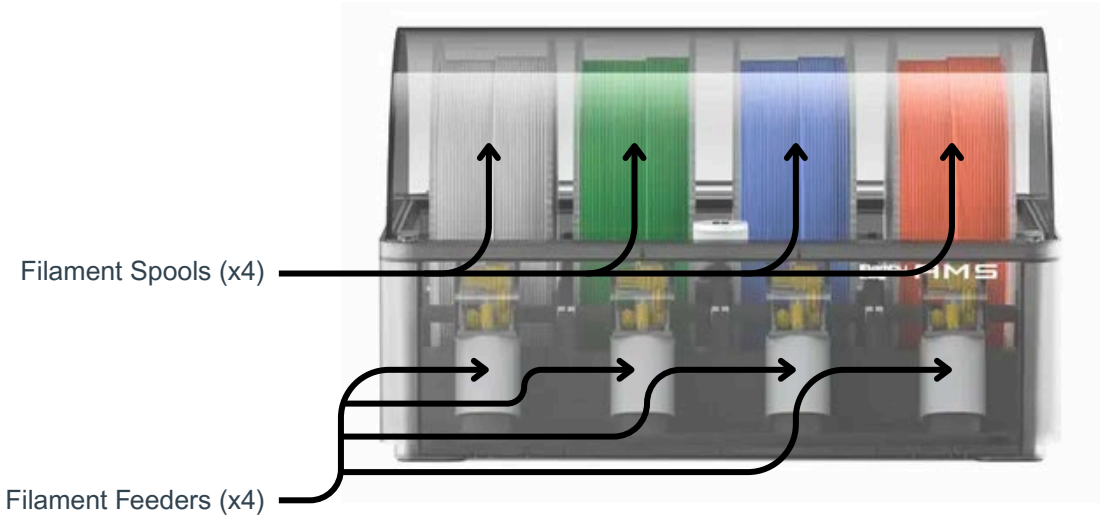


Photo Credit: Bambu Labs
https://ca.store.bambulab.com/products/amsmulticolorprintinggad_source=1&gad_campaignid=21815873208&gbraid=0AAAAA9pdpI4AAW2FQAd2Z3PX507vh7BbO&gclid=CjwKCAjwy7HEBhBJEiwA5hQNohmQuR6Cciu62lkyscLCp4oupwO-dMeYOEuKrQIBs1NnQnrb2snkbRoC0U0QAvD_BwE

Checklist for Use

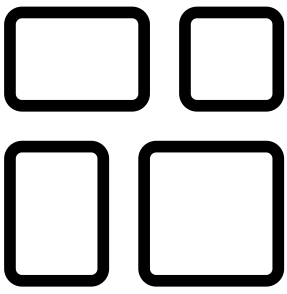
Follow this helpful overview of steps

6 Easy Steps to 3D printing from home

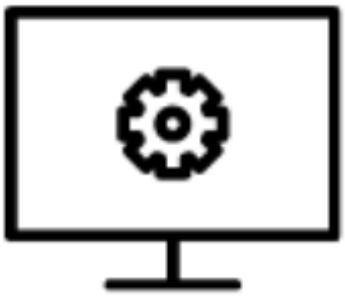
- [ ] **1. Find**
  Find a file to print through sites like thingiverse or create your own

- [ ] **2. Log In**
  Log into 3DPrinterOS

- [ ] **3. Upload**
  Upload your .stl file



- [ ] **4. Layout**
  Adjust file scale, orientation and placement



- [ ] **5. Slice**
  Select a profile to auto prep your file for printing

- [ ] **6. Send**
  Select your printer to start printing!

For more detailed instructions continue to pg.11

Beginning to 3D Print

1. Find

a. Search for an object to print on Thingiverse, Thangs, or another 3D model website.

b. Download the .stl file (can also download obj. file).

c. Alternatively, if using your own model from software such as Tinkercad or Blender make sure to download your design as an stl. file.

d. If you prefer to use your own slicing software like Cura you can upload your gcode file as long as you have sliced it for one of our available printers.

2. Log In.

Use your MacEwan username and password on our 3D Printing page, 3DPrinterOS.

3. Upload

a. On the Files tab - click Add Files.

b. Select your chosen .stl, .obj. or .gcode file.

Beginning to 3D Print

4. Layout

If your print is already the right scale, placement and orientation on the plate skip this step and go straight to slicing (pg. 14) Orientation can be adjusted in the slicing step if needed.

a. Once file is uploaded select the layout function. 

b. Your file will open up in an editing window. Here you can adjust the scale, placement and orientation of your print. Before starting make sure to select the right printer from the drop down printer tab.

c. To rotate the 3D space (not the object):

   i. Touchpad: Click with your pointer finger and slide your middle finger on the pad.

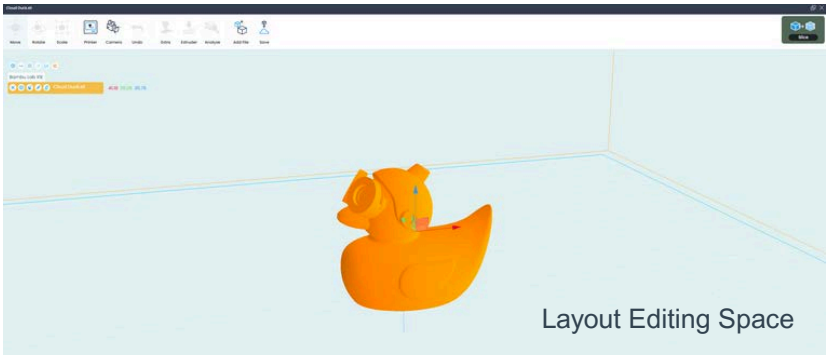
   ii. Mouse: Hold both left and right buttons and move the mouse.

d. To enlarge or shrink the 3D space (not the object):

   i. Touchpad: Use two fingers to either push out or come together

   ii. Mouse: Use the scroll wheel

e. Alternatively you can also navigate using the drop down menu from the camera tab and select a viewing angle.

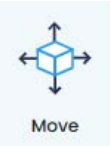


Layout Editing Space

Beginning to 3D Print

4. Layout (continued)

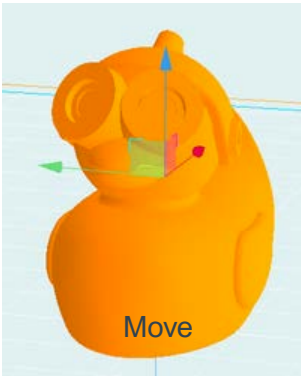
f. Move the object:



i. Select the object, then open the Move tab. (when you enter layout mode the Move tab will be selected.

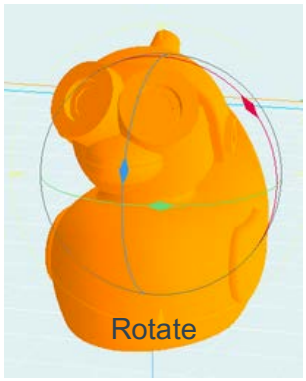
ii. Select Optimal Rotate, Center, and On bed to ensure the object is in the best printing orientation, is centered and is touching the build plate.

iii. If more movement is needed, use the arrows to slide it along the X, Y, or Z axis, or enter specific coordinates.



g. Rotate the object:

i. Select the object, then open the Rotate tab. Use the circular guides to turn it on any axis, or enter angles in degrees.



h. Scale the object (To scale one axis only, toggle “keep proportions”):

i. Select the object, then open the Scale tab.

Free scale: Drag the cubes on each axis.

Exact size: Enter a measurement in mm — other axes adjust automatically if “keep proportions” is selected.

By percentage: Enter a percent value and press Enter. - other axes adjust automatically if “keep proportions” is selected.

Beginning to 3D Print

5. Slice

a. Select the slice function:

i. If in the main file directory it is listed alongside each file 

ii. If in the layout mode it is in the top right hand corner 

b. Select 3D Printer (see pg. 3&4 for printer specifics):

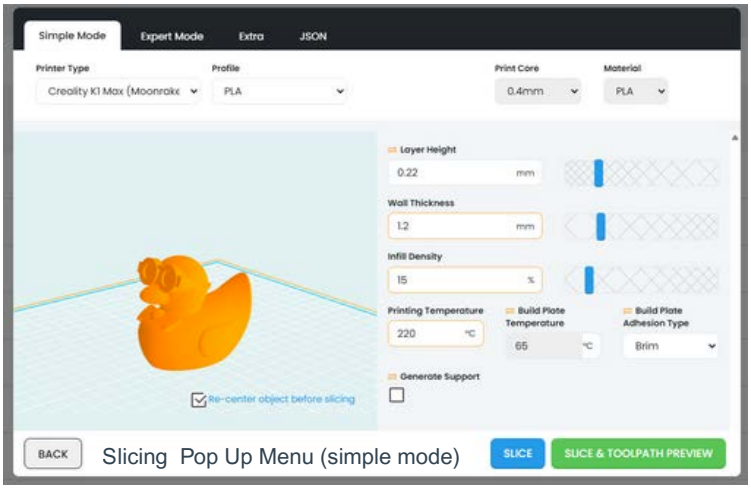
i. If in the main file directory you can select the printer within the slicer window using the printer drop down menu. 

ii. If in the layout mode make sure the correct printer is selected from the printer tab before slicing. 

c. Determine between two preset slicing profiles:

i. No support - for when you do not have any unsupported overhangs in your object (pg.19).

ii. Support - adds in support that can be removed after printing (pg.19).



Slicing Pop Up Menu (simple mode)

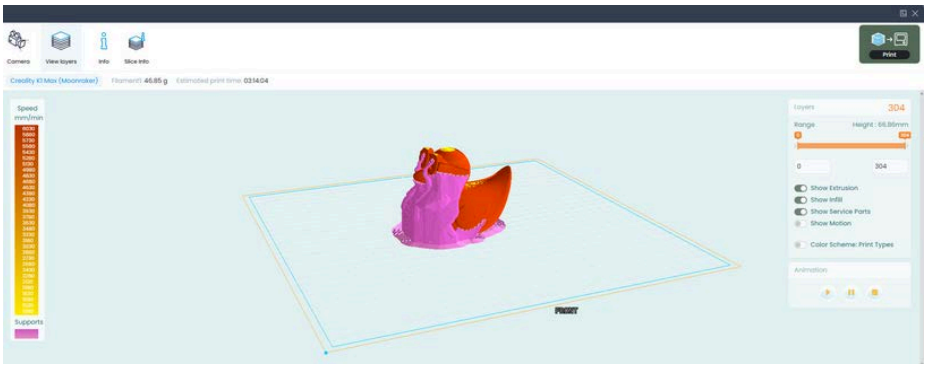
Beginning to 3D Print

5. Slice Continued

d. Select Slice & Toolpath Preview. 

e. Review your file for any problems, see what the supports will look like, and optionally review the print layer by layer using the slider in the right hand menu.

f. Review the estimate for the print time and optionally the amount of filament used. Ensure the time is under the limit of 12hrs.



6. Print

a. Select Print. 

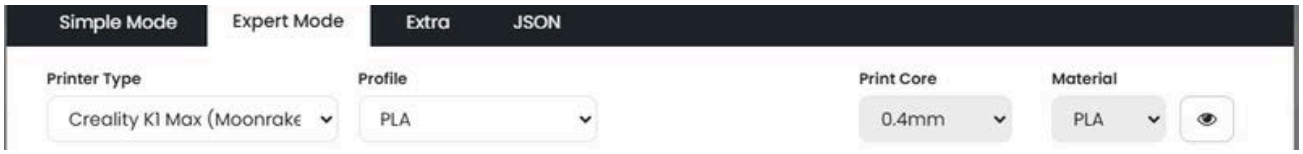
b. In the "Add a note" section please specify the colour you would like to print your object in. We have a list of regularly stocked colours on our website.

c. Choose a printer. Note the current queue time for each printer.

d. Click Queue to add your job to the list. Once we start your job you will receive a notification by email. 

Advanced Slicing

If you are interested in learning more advanced 3D printing you can modify settings in the 3DPrinterOS slicer using the expert mode or use an external slicer like Cura.



Slicing Settings

Setting	Definition
Support	Structures that are needed when parts of your print would otherwise be in mid-air (overhangs over ~45°).
Support Placement	Two types • Everywhere = builds supports even on top of the model. • Touching Buildplate = only builds supports from the bed up.
Support Pattern	Affects how easy the support is to remove and how well it holds up overhangs. • Trees are a common pattern and are in the Makerspace presets.
Cooling Fan	Cools down filament quickly after extrusion for better print quality.
Print Speed	How fast the printer moves and lays down filament. Limited by machine. • Slower = better quality, especially for small details or bridges. • Faster = shorter print times, but may lower quality.

Advanced Slicing

Slicing Settings Continued

Setting	Definition
Layer Height	Controls how thick each layer of your print is. • Lower values (e.g., 0.1mm) = smoother, more detailed prints but take longer. • Higher values (e.g., 0.2mm) = faster prints, less detail. Note: do not go less than 1 mm - 2 mm if it is a larger object.
Initial Layer Height	Usually thicker to help the print stick better to the build plate. • This can affect the look of the bottom of your print.
Shell	Refers to the outer parts of your print: the walls, top, and bottom. • Wall Thickness - thicker walls = stronger prints. • Top/Bottom Thickness - Prevents gaps and improves the solid look and feel of the print.
Infill	The internal structure of the print that gives it strength. No infill = hollow.
Infill Density	The percentage of internal structure printed. • Average (10–20%) = for most decorative prints. • Higher (30–100%) = for strong, functional parts. Typically no need to go beyond 50%.
Infill Pattern	Changes how the inside is filled (e.g., grid, honeycomb). Affects strength and print time.
Travel Speed	How fast the nozzle moves when not printing (just moving between areas).
Retraction	Pulling back filament to prevent oozing during travel moves.

Advanced Slicing

Slicing Settings Continued	
Setting	**Definition**
Build Plate Adhesion	Helps the print stick to the plate and prevents warping or detaching. 3 main types - skirt, brim and raft.
Skirt	A line around the object to get the nozzle ready (no contact with the object).
Brim	A flat ring connected to the first layer to help with grip (useful for small parts).
Raft	A thick base layer under the object to improve adhesion (great for tricky prints or warped materials).

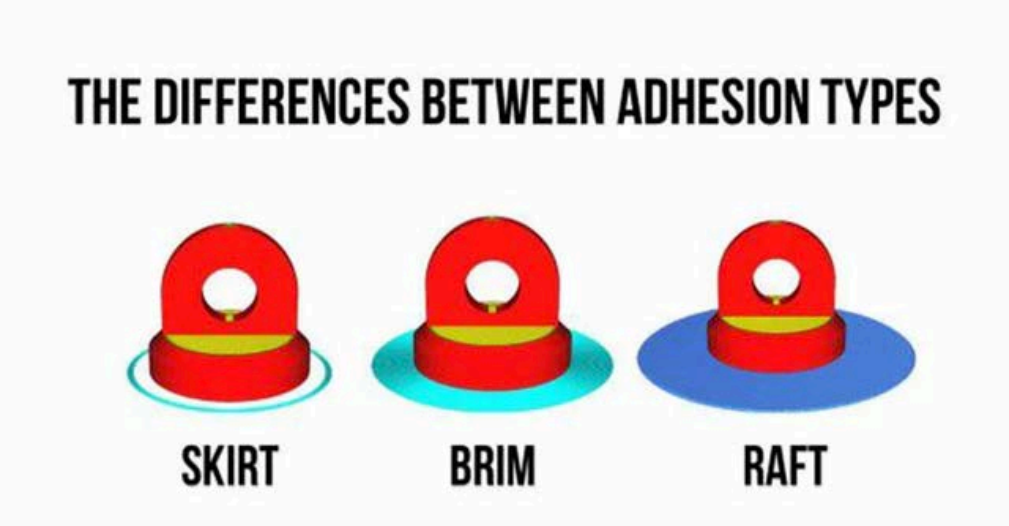


Photo Credit: Kingroon
https://kingroon.com/blogs/3dprintingguides/howtouseraftsbrimsandskirtsin3dprintingsrsltid=AfmBOoqtS_xhpsfmjaPQPxwEdRMko_c15NwRymv-ft3NdS2hQjIlpDNS

Troubleshooting

Common Issues		
Problem	**Likely Cause**	**Fix**
Unintended gaps or holes in print	Model not watertight.	Use a repair tool before submitting.
Missing supports	Model has steep angles (>45°).	Re-slice with supports included.
Missing small details	Parts are smaller than the printer's resolution.	Increase the size of fine features or the overall model.
Print breaks along layer lines	Print is weakest along layer lines so small details are more fragile.	Change orientation of model and re-slice.
Nothing printed / failed early	File had errors.	Try re-slicing and re-submitting file. If issues continue seek help from tech tutors.

Prints with overhangs without support

Photo Credit: Baptiste Higgs, Medium
https://medium.com/bravovictornovember/3d-print-overhangs-and-how-to-deal-with-them-9eed6a7bcb5d

Prints with overhangs with support

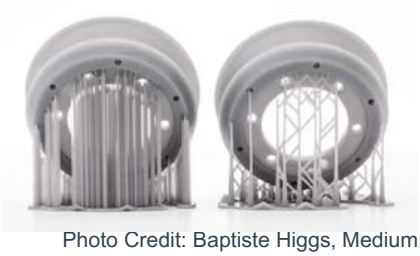


Photo Credit: Baptiste Higgs, Medium
https://medium.com/bravovictornovember/3dprintoverhangs-and-how-to-deal-with-them-9eed6a7bcb5d

References

3DPrinterOS. (n.d.). 3D printer management software: 3D printer operating system. 3D Printer Management Software | 3D Printer Operating System. https://www.3dprinteros.com/

I acknowledge the use of ChatGPT (https://chat.openai.com/) to help develop this instructional guide.

3D Printing Guide © 2025 by MacEwan University Makerspace is licensed under CC BY-NC 4.0. To view a copy of this license, visit https://creativecommons.org/licenses/by-nc/4.0/