

LULZBOT TAZ PRO PRINTER

STUDENT INNOVATION CENTER
ROOM 2222

Wade Warman
Dayon Royster

VERSION 2.0

10/9/2020

OVERVIEW.....1

HEALTH & SAFETY INFO.....2

HAZARD CONTROL MEASURES/REQUIRED PPE.....3

FIRST AID PROCEDURES.....4

WASTE DISPOSAL.....5

SPILL/CLEAN UP PROCEDURES.....6

OPERATIONS GUIDE.....7

 SOFTWARE INTERFACE8

 SOFTWARE OPERATION (SINGLE EXTRUDER)11

 SOFTWARE OPERATION (DUAL EXTRUDER).....24

 MACHINE OPERATION41

 AFTER PRINTING.....48

OVERVIEW

The purpose of this document is to provide standard operating procedures for the use of the Taz Pro printers in room 2222 of the Student Innovation Center

HEALTH & SAFETY INFO

Nanoparticle Emissions: Nanoparticles (ultrafine particles less than 1/10,000 of a millimeter) are one of the byproducts emitted during the 3D printing process. Nanoparticles are of concern because they are very small, have large surface areas and can interact with the body's systems, including the skin, lungs, nerves and brain.

Chemical Vapors: Heating of certain thermoplastic filament can generate toxic vapors and vapors with high volatile organic compounds (VOCs).

Hot Surfaces: The print head block and UV lamp generate heat. Such surfaces must be guarded and labeling must warn users of the hazards.

Mechanical Hazards: Moving parts must be guarded to prevent accidental contact. Guards must never be bypassed.

Ultraviolet Light (UV): UV light poses a radiation hazard that can cause damage to vision and skin. Do not look at the lamp and ensure the UV screen is intact before use.

Electrical: Contact with energized parts can lead to injury or even death. Before each use, inspect the 3D printer for any damaged wiring and safeguards. Do not use the printer if problems are found. Ensure the printer is properly grounded and plugged directly into an outlet.

HAZARD CONTROL MEASURES AND REQUIRED PPE

REQUIRED PPE:

- Safety glasses
- Closed toed shoes

Hazard Control Measures:

- Wait until the printer is completely finished printing before removing parts/material
- Tie long hair back
- The build platform and extruders are HOT!!! They can burn you. Do not touch them until they have had a chance to cool to room temperature
- Only print with PLA filament. If you are unsure or want to experiment with non-PLA materials, please ask a Shop Supervisor, we would be happy to help figure it out!
- The scraper used to remove prints is sharp! Be very careful and ONLY scrape away from your body!
- Wash your hands after using the printers, especially SLA (resin) printers

FIRST AID PROCEDURES

BURNS: Minor burns are typically small, red, have swelling, and can blister. Cool burns with cold water and continue until the pain lessens. After cooling, cover with a dry, sterile bandage or clean dressing. Consult a physician as needed.

CUTS/SCRAPES: Minor cuts and scrapes usually stop bleeding on their own. If needed, apply gentle pressure with a clean bandage or cloth and elevate the wound until bleeding stops. Clean the wound and apply bandages. Consult a physician as needed.

PINCH/CRUSH: Consult a physician as needed.

INHALATION: Ensure there is proper ventilation prior to use. If needed, stop the procedure and move to a well ventilated area. Consult a physician as needed.

EYES: Immediately irrigate the eyes at an eyewash station for at least 15 minutes. Hold the eyelids away from the eyeball, moving eye in all directions to wash thoroughly behind the eyelids. Consult a physician as needed.

All accidents and injuries occurring at work or in the course of employment must be reported to the employee's supervisor as soon as possible (even if no medical attention is required).

<http://www.ehs.iastate.edu/occupational/accidents-injuries>

WASTE DISPOSAL

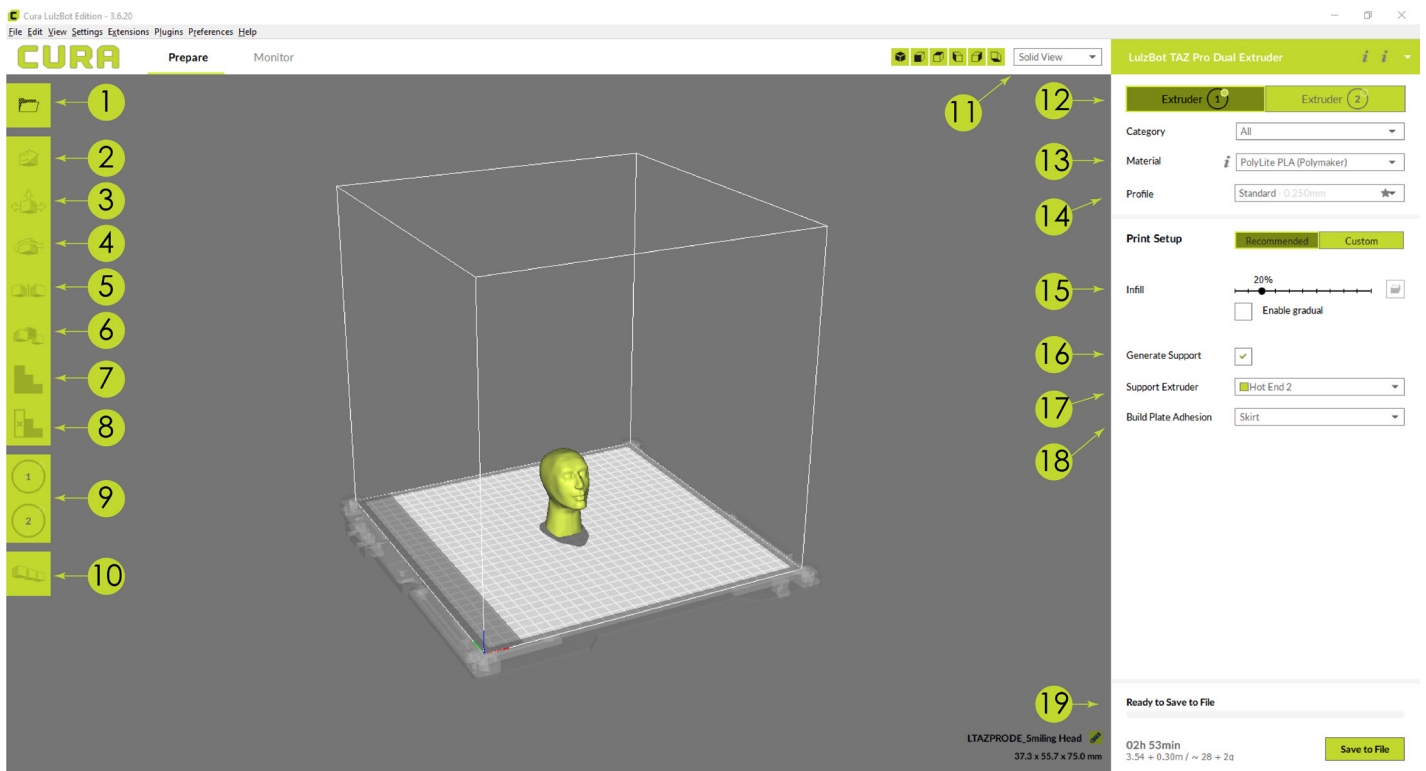
For all failed prints, support structures, or excess filament, allow both the machine and material to cool to room temperature first then dispose of in the bin labeled "PLA RECYCLING". We donate our PLA waste to the CCL in College of Design.

SPILL/CLEAN UP PROCEDURES

Allow both the machine and PLA material to cool to room temperature first then dispose of in the bin labeled "PLA RECYCLING".

OPERATIONS GUIDE

SOFTWARE INTERFACE



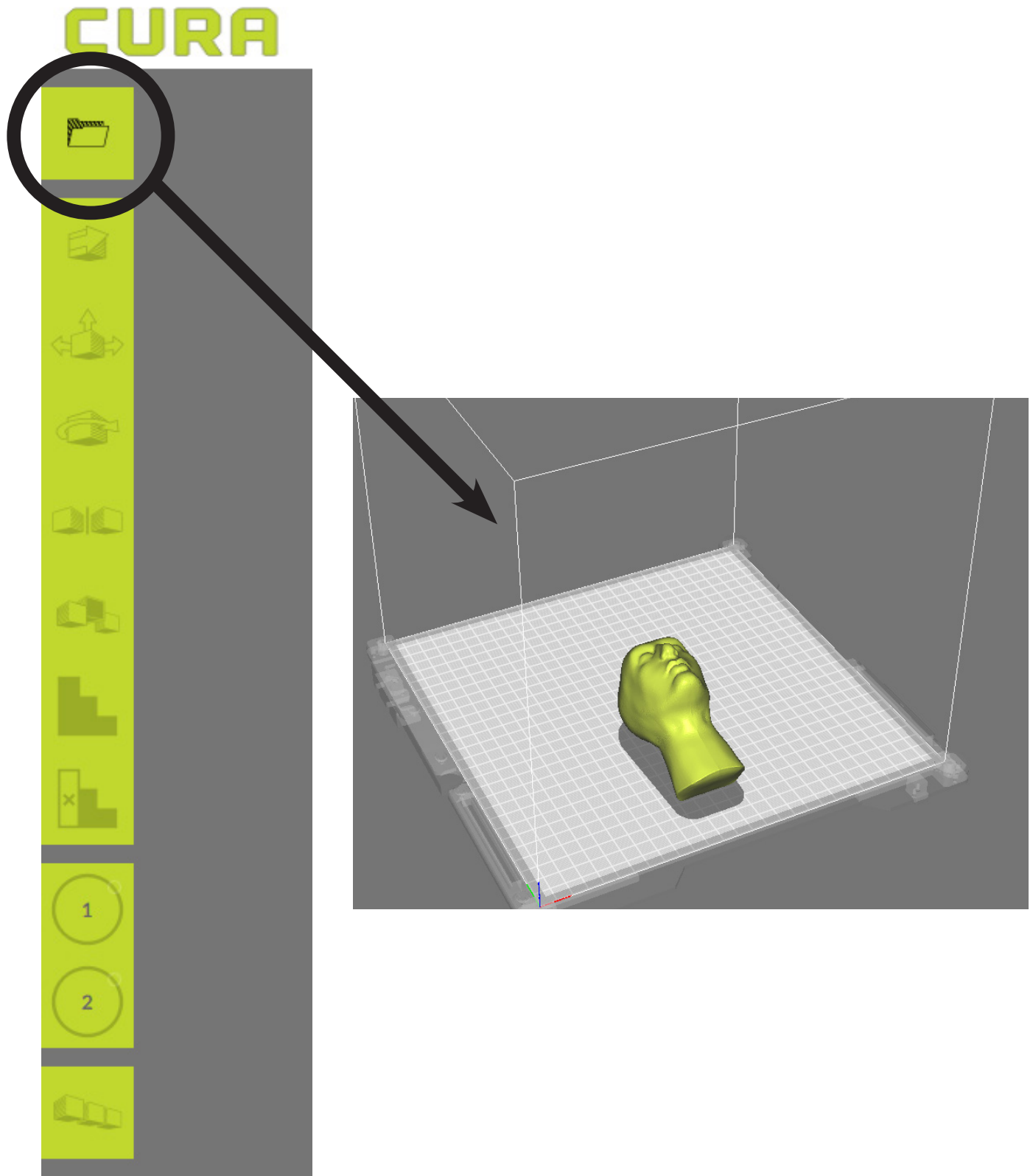
- 1. Open File:** Import the file to be printed.
- 2. Move Model:** Move the model
- 3. Scale Model:** Adjust the size of the model. (NOTE: Cura uses metric, not imperial)
- 4. Rotate Model:** Rotate the model via X, Y, Z axis
- 5. Mirror Model:** Creates a reverse/mirrored image of the model selected in either the X, Y, or Z axis directions.
- 6. Per Model Settings:** Allows you to print two or more objects with different slicing settings between the two.
- 7. Custom Supports:** Allows you to create your own support structures
- 8. Support Blocker:** Allows you to block certain areas of the model from having support structures added
- 9. Extruder Assignment:** This allows you to choose which extruder you wish to use for your print.
- 10. Multiply Object:** Create multiples of your model to be printed
- 11. View Options:** Solid view = the standard view and shows the solid outer surfaces of the model. X-Ray View = allows you to look inside of the object, Layer View = allows you to see the print, layer by layer and also to view the generated support structures
- 12. Extruder:** Assign which extruder to print with
- 13. Material Type:** What kind of material to print with
- 14. Profile:** Resolution of the print. *High Detail* = better surface finish but longer print times, *Standard* = decent surface finish, decent print time, *High Speed* = faster print time, but more coarse surface finish

- 15. Infill:** How solid the interior of the print is in percentage
- 16. Generate Support:** Tells the printer to create support structures where needed (default is any surface equal or greater than 45 degrees)
- 17. Support Extruder:** Assign which extruder to build support structures with.
- 18. Build Plate Adhesion:** Used to select different methods for helping ensure your model stays adhered to the build plate, and the extruder is prepared for printing. *Skirt* = This prepares the extruder to ensure filament is flowing properly before actually beginning to print your part. *Brim* = Brim creates a single layer attached to and protruding around the base of your object. This increases the surface area of your print, preventing warping and ensuring adhesion to the build plate. *Raft* = The raft option will build up a layer of material underneath your object as a "platform" for the printed object to rest on.
- 19. Save to File/Save to Removable Drive:** Allows you to save your prepared file to either your computer or a USB drive.

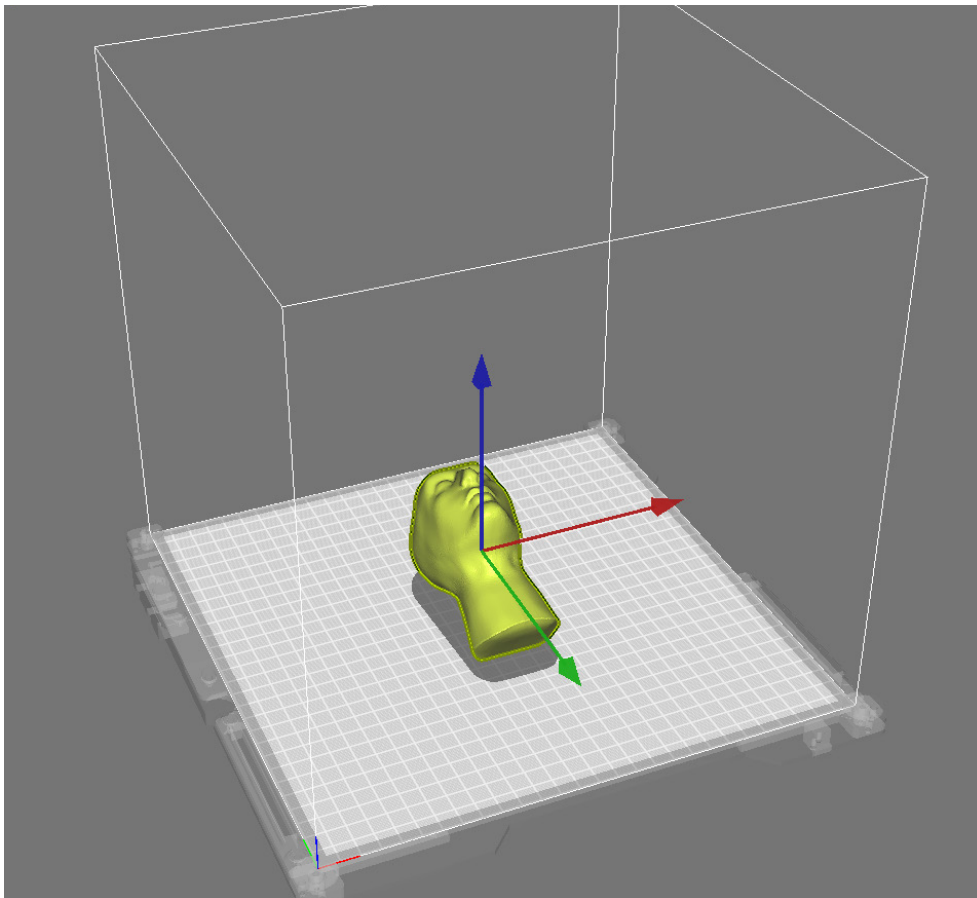
SOFTWARE OPERATION (SINGLE EXTRUDER)

The following steps are for using a single color/material only. If you are using two colors of filament or two different material types, go to the section titled “DUAL EXTRUDER” on page 17.

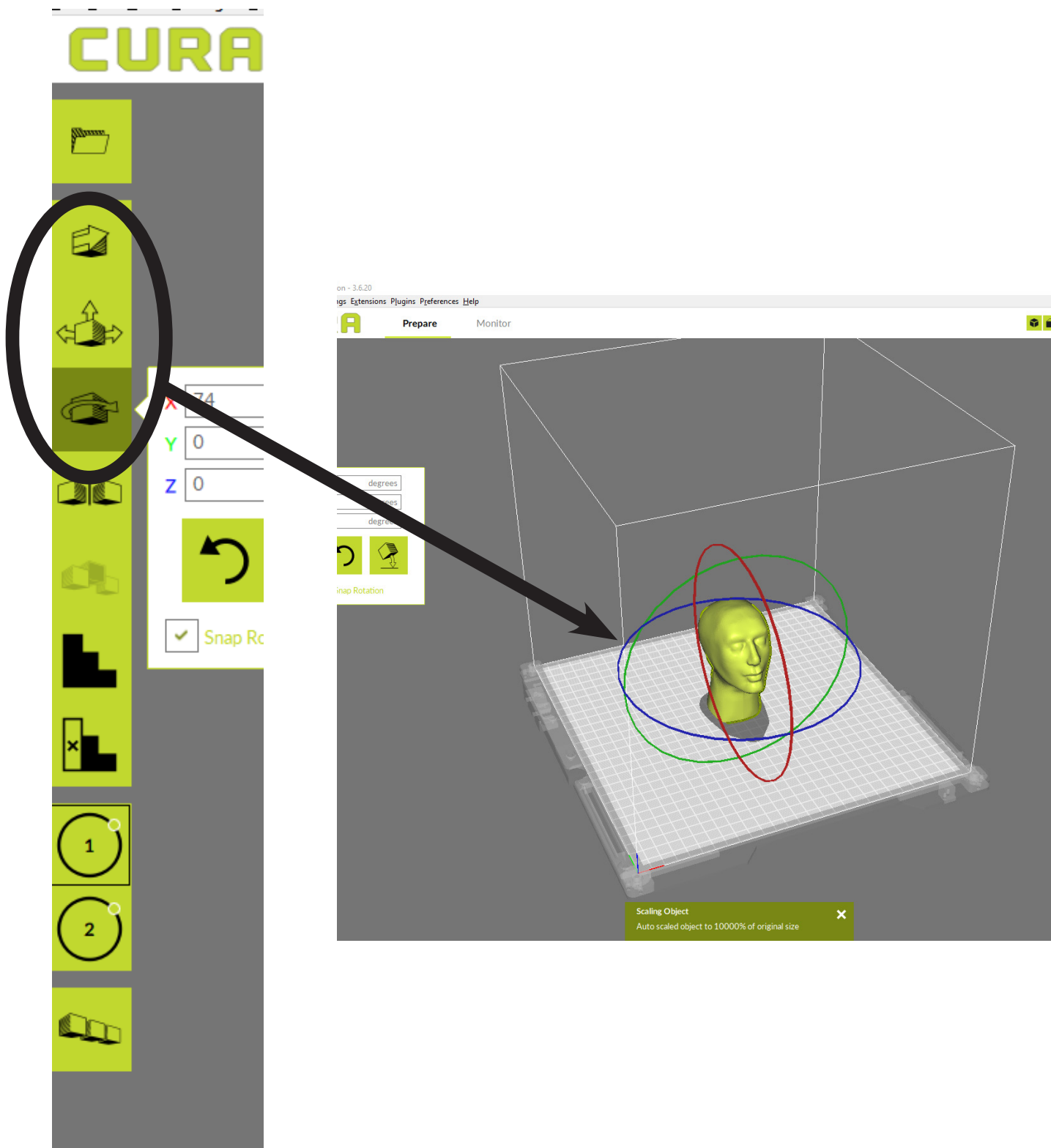
1. Click the “Open File” icon. In the dialog window, locate your file and click “Open”. Your model will appear on the virtual build platform.



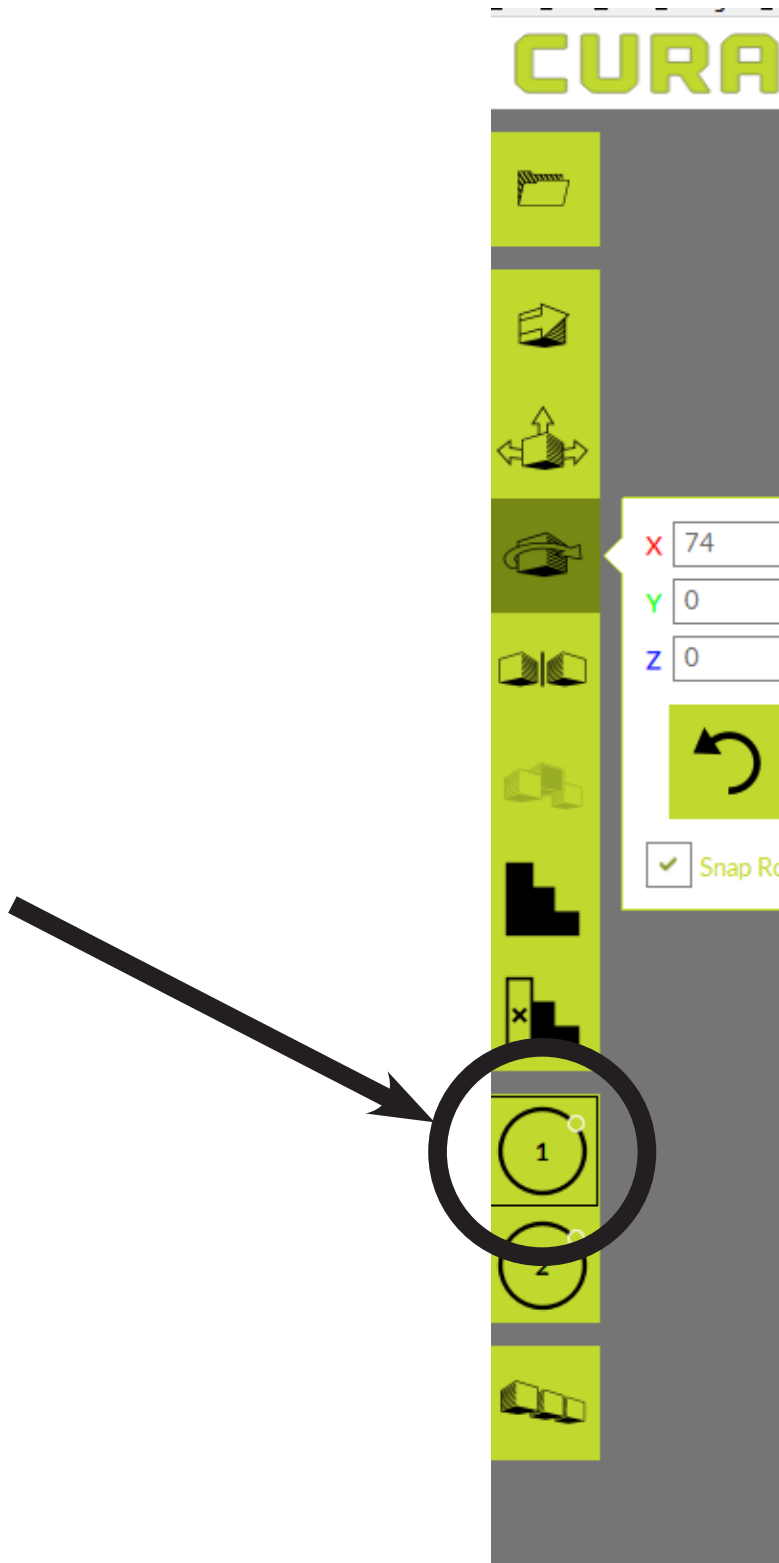
2. Click your model to select it. Directional arrows will appear on the object. (NOTE: it may import as the wrong size as Cura uses metric).



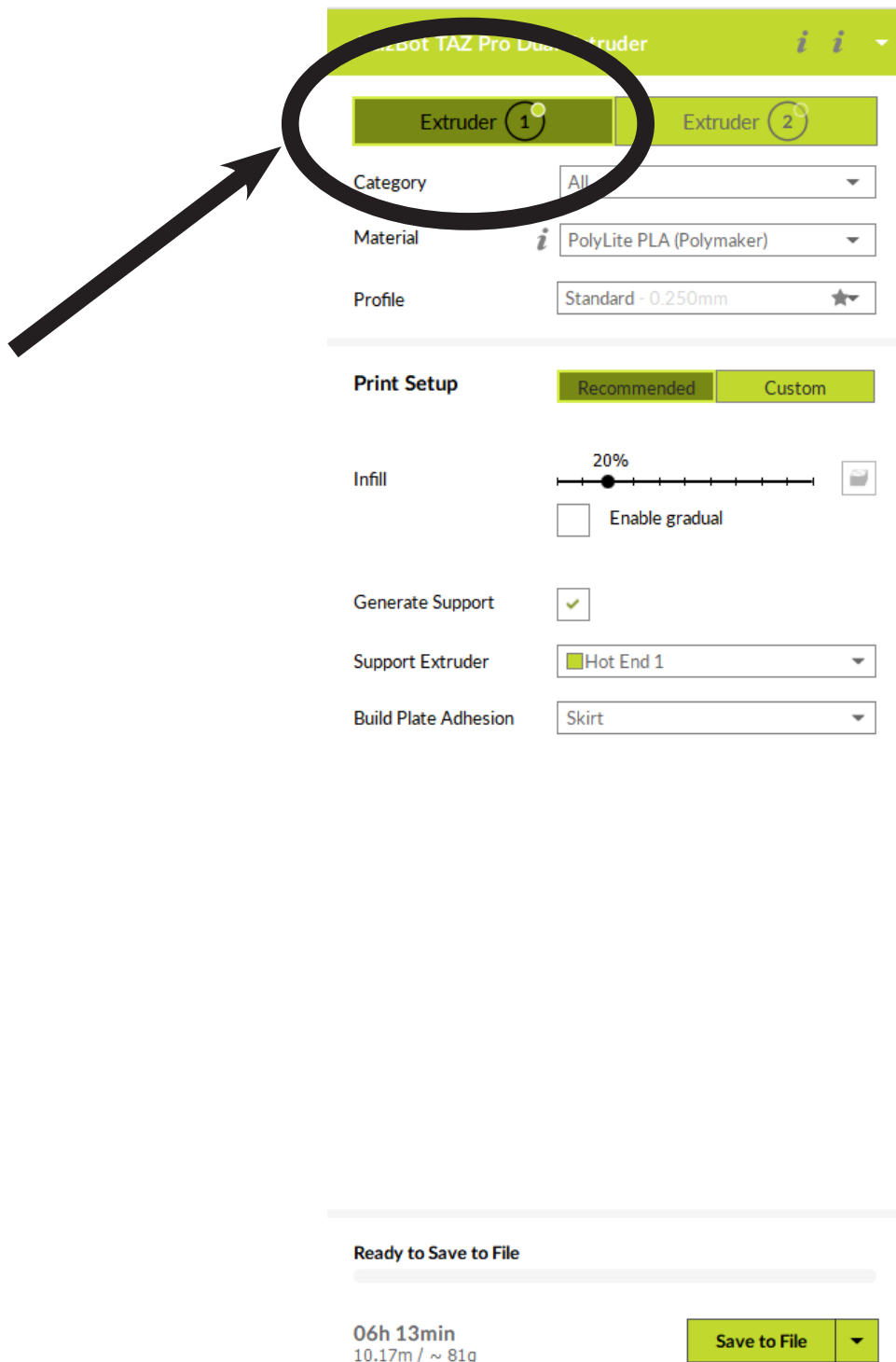
3. If needed, use the icons on the left toolbar to adjust your model (Scale, Rotate, Move, etc.)



4. Choose the “Hot End” you wish to use via the left toolbar icon (1) or (2). (The icon will have a box around your selection).



5. In the right toolbar, make sure the extruder selected matches the Hot End you selected in step #4. (i.e. Hot End 1 = Extruder 1)



The screenshot displays the software's configuration panel for a printer. At the top, a green header bar contains the printer name 'Bot TAZ Pro Dual Extruder' and an information icon. Below this, two buttons labeled 'Extruder 1' and 'Extruder 2' are shown. 'Extruder 1' is highlighted with a black circle, and a black arrow points to it from the left. Below the extruder buttons are several configuration options: 'Category' (set to 'All'), 'Material' (set to 'PolyLite PLA (Polymaker)'), and 'Profile' (set to 'Standard - 0.250mm'). A 'Print Setup' section has 'Recommended' selected over 'Custom'. The 'Infill' section shows a slider at 20% and an unchecked 'Enable gradual' checkbox. 'Generate Support' is checked. 'Support Extruder' is set to 'Hot End 1'. 'Build Plate Adhesion' is set to 'Skirt'. At the bottom, a 'Ready to Save to File' status bar shows a progress indicator, a time estimate of '06h 13min' and '10.17m / ~ 81g', and a 'Save to File' button.

6. In the drop-down labeled “Material”, make sure it says “PolyLite PLA (Polymaker)” (NOTE: If you want to use a material other than PLA, please speak to one of the Shop Supervisors)

The screenshot displays the LulzBot TAZ Pro Dual Extruder software interface. At the top, there is a green header with the text "LulzBot TAZ Pro Dual Extruder" and a user profile icon. Below this, there are two tabs for "Extruder 1" and "Extruder 2". The "Material" dropdown menu is highlighted with a black circle and contains the text "PolyLite PLA (Polymaker)". A large black arrow points from the left towards the "Material" dropdown. Below the material selection, there are several settings: "Category" (All), "Profile" (Standard 0.250mm), "Print Setup" (Recommended/Custom), "Infill" (20% with a slider and "Enable gradual" checkbox), "Generate Support" (checked), "Support Extruder" (Hot End 1), and "Build Plate Adhesion" (Skirt). At the bottom, there is a "Ready to Save to File" section with a progress bar, a time estimate of "06h 13min" and "10.17m / ~ 81g", and a "Save to File" button.

7. In the drop-down menu labeled “Profile” you can adjust the resolution of your print. Keep in mind the trade-offs: High Detail = longer print time, but better surface finish; High Speed = Less resolution but a faster print time

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
 Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

8. If needed, adjust your infill. The default setting is 20%, which is fine for most prints. However, again there is a trade-off: Higher Infill Density = stronger print, but slower print time; Lower Infill Density = Weaker print (easier to damage) but faster print time

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
 Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

9. If your model has over-hangs, make sure to click the “Generate Support” check-box. (NOTE: this will give you the option to use the other Hot End/Extruder to print the support structure. If you wish, you can simply use the same Hot End/ Extruder to print both the model and the support structure.)

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

10. In the drop-down menu labeled “Build Plate Adhesion”, select the style of adhesion you want. (NOTE: It is NOT recommended to select “NONE”, as this increases the likelihood of a failed print due to the model “peeling” away from the build platform during the print.

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
 Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

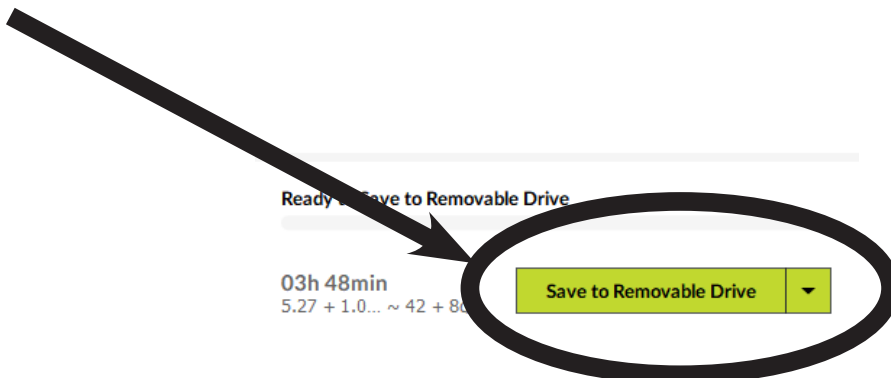
Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

11. If you are finished making all adjustments to both the model and print settings, insert a USB drive into your computer and click the “Save to Removable Drive” button in the lower right-hand part of the screen. (NOTE: You may need to click the arrow to change the selection from “Save to File” to “Save to Removable Drive”)

The screenshot shows the LulzBot TAZ Pro Dual Extruder software interface. At the top, there is a green header with the text "LulzBot TAZ Pro Dual Extruder" and two information icons. Below the header, there are two tabs for "Extruder 1" and "Extruder 2". The main settings area includes a "Category" dropdown set to "All", a "Material" dropdown set to "PolyLite PLA (Polymaker)", and a "Profile" dropdown set to "Standard - 0.250mm". Below this is a "Print Setup" section with two tabs: "Recommended" (selected) and "Custom". The "Infill" section shows a slider set to 20% and an unchecked "Enable gradual" checkbox. The "Generate Support" checkbox is checked. The "Support Extruder" dropdown is set to "Hot End 1", and the "Build Plate Adhesion" dropdown is set to "Skirt".



12. Remove your USB from your computer and insert it into the Taz Pro

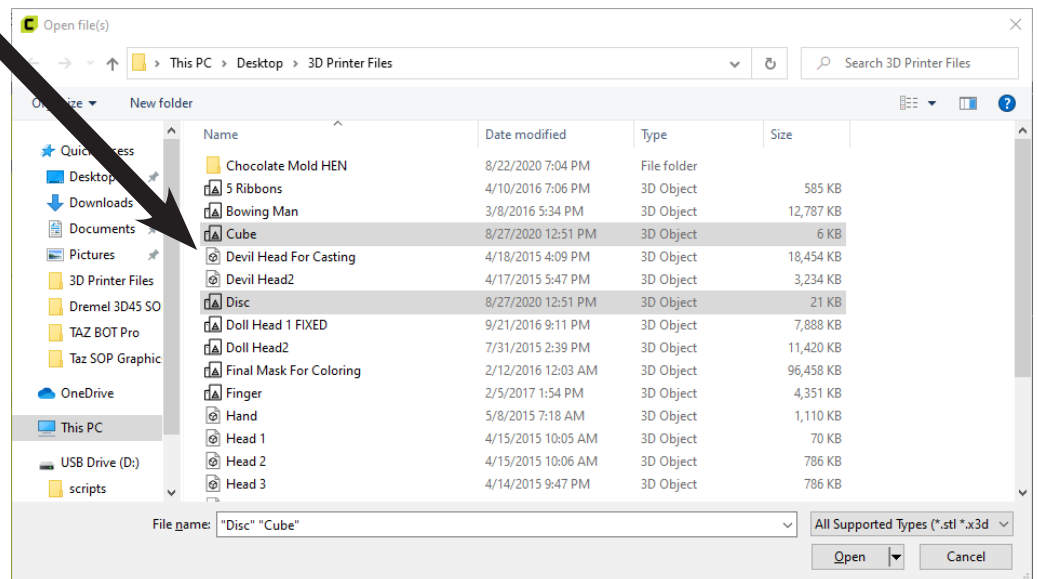


SOFTWARE OPERATION (DUAL EXTRUDER)

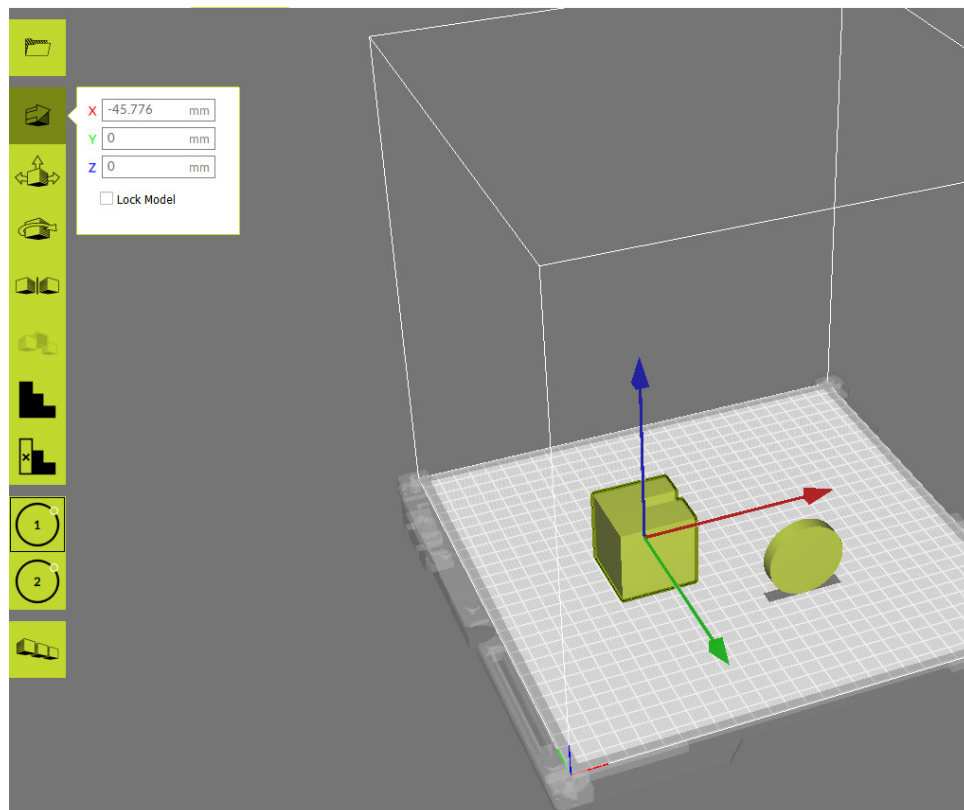
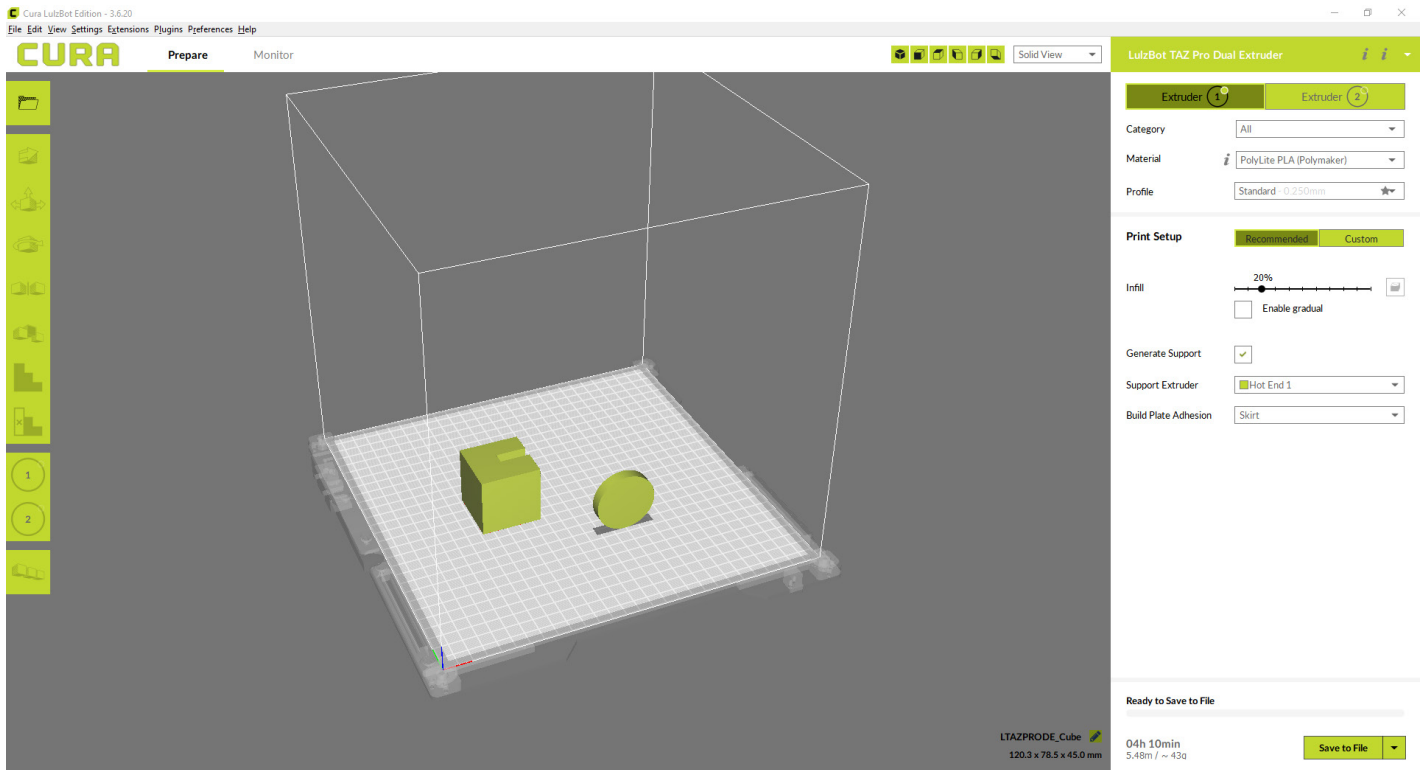
The following steps are for using multiple colors/materials. If you are using only one color of filament or one material type, go to the section titled "SINGLE EXTRUDER" on page 04.

NOTE: Dual extrusion is the process of 3D printing with multiple filaments. You can mix colors or different materials with a print head that has two extruders and nozzles. With two spools loaded, the printer alternates between them by printing one at a time. It's not actually faster at printing because its still using only one extruder at a time.

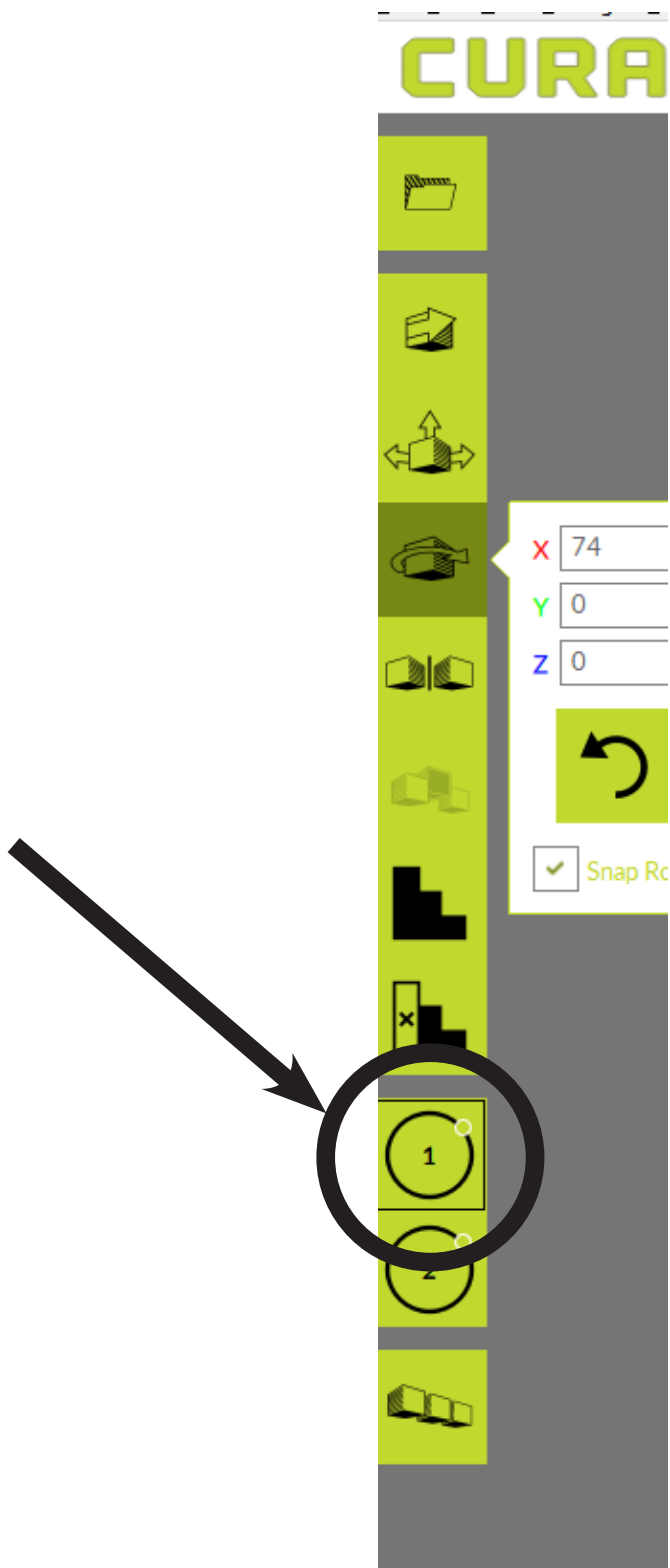
1. Click the “Open File” icon on the right toolbar. In the dialog window, locate your file/files and click “Open”



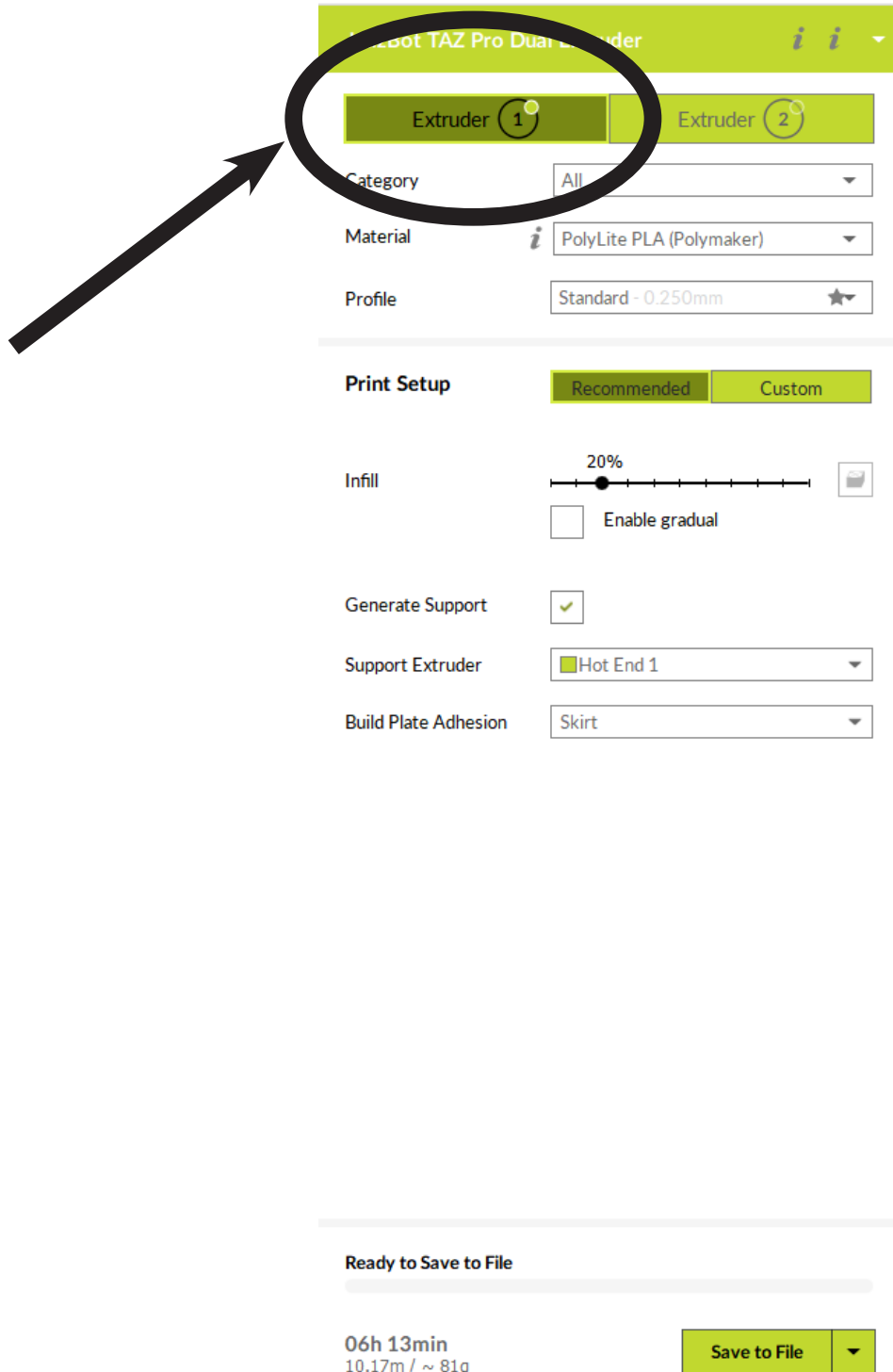
2. You should see your models in the virtual build platform. Click on the first model that you want to assign Hot End/Extruder 1. The Model will be highlighted when you do so.



3. On the left toolbar, select the Hot End you wish to use for this component of your print. Your Hot End selection will have a box around it.



4. In the right toolbar, make sure that the Extruder selection corresponds to the Hot End you selected in step #3.



The screenshot displays the software interface for a printer, specifically the 'Extruder' selection section. At the top, there are two buttons labeled 'Extruder 1' and 'Extruder 2'. The 'Extruder 1' button is circled in black, and a large black arrow points to it from the left. Below these buttons, there are several dropdown menus: 'Category' (set to 'All'), 'Material' (set to 'PolyLite PLA (Polymaker)'), and 'Profile' (set to 'Standard - 0.250mm').

Below the dropdowns, there is a 'Print Setup' section with two tabs: 'Recommended' (selected) and 'Custom'. Under 'Recommended', there is an 'Infill' slider set to 20%, an 'Enable gradual' checkbox (unchecked), a 'Generate Support' checkbox (checked), a 'Support Extruder' dropdown menu (set to 'Hot End 1'), and a 'Build Plate Adhesion' dropdown menu (set to 'Skirt').

At the bottom of the interface, there is a 'Ready to Save to File' status bar. Below this, the estimated print time is shown as '06h 13min' and '10.17m / ~ 81g'. To the right of this information is a 'Save to File' button with a dropdown arrow.

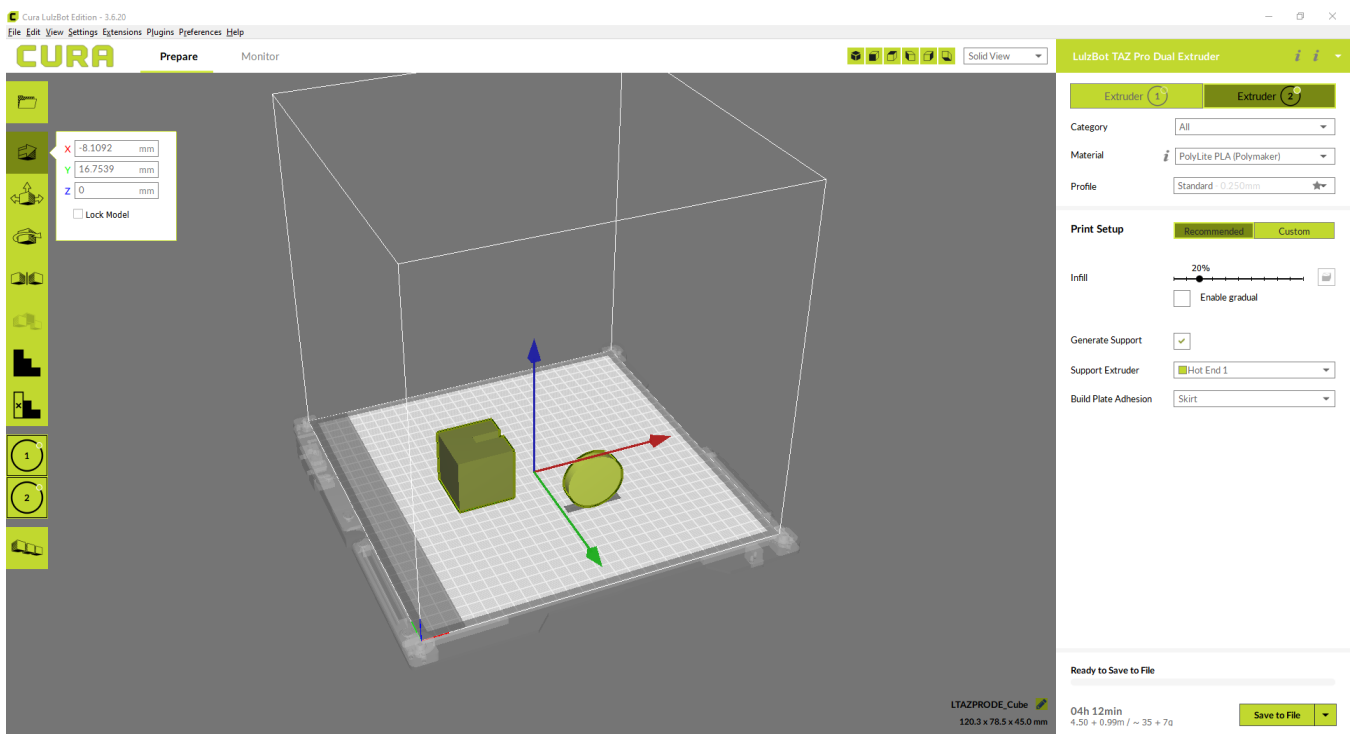
5. Make sure you select the correct material from the “Material” drop-down menu in the right toolbar

The screenshot displays the software interface for a LulzBot TAZ Pro Dual Extruder. At the top, there are two extruder selection buttons labeled "Extruder 1" and "Extruder 2". Below these are three dropdown menus: "Category" (set to "All"), "Material" (set to "PolyLite PLA (Polymaker)", which is circled in black), and "Profile" (set to "Standard 2850mm"). A large black arrow points from the left towards the "Material" dropdown. The "Print Setup" section includes two tabs: "Recommended" (selected) and "Custom". Under "Recommended", there is an "Infill" slider set to 20%, an unchecked "Enable gradual" checkbox, a checked "Generate Support" checkbox, a "Support Extruder" dropdown set to "Hot End 1", and a "Build Plate Adhesion" dropdown set to "Skirt". At the bottom, a status bar shows "Ready to Save to File", a progress indicator, and a "Save to File" button with a dropdown arrow.

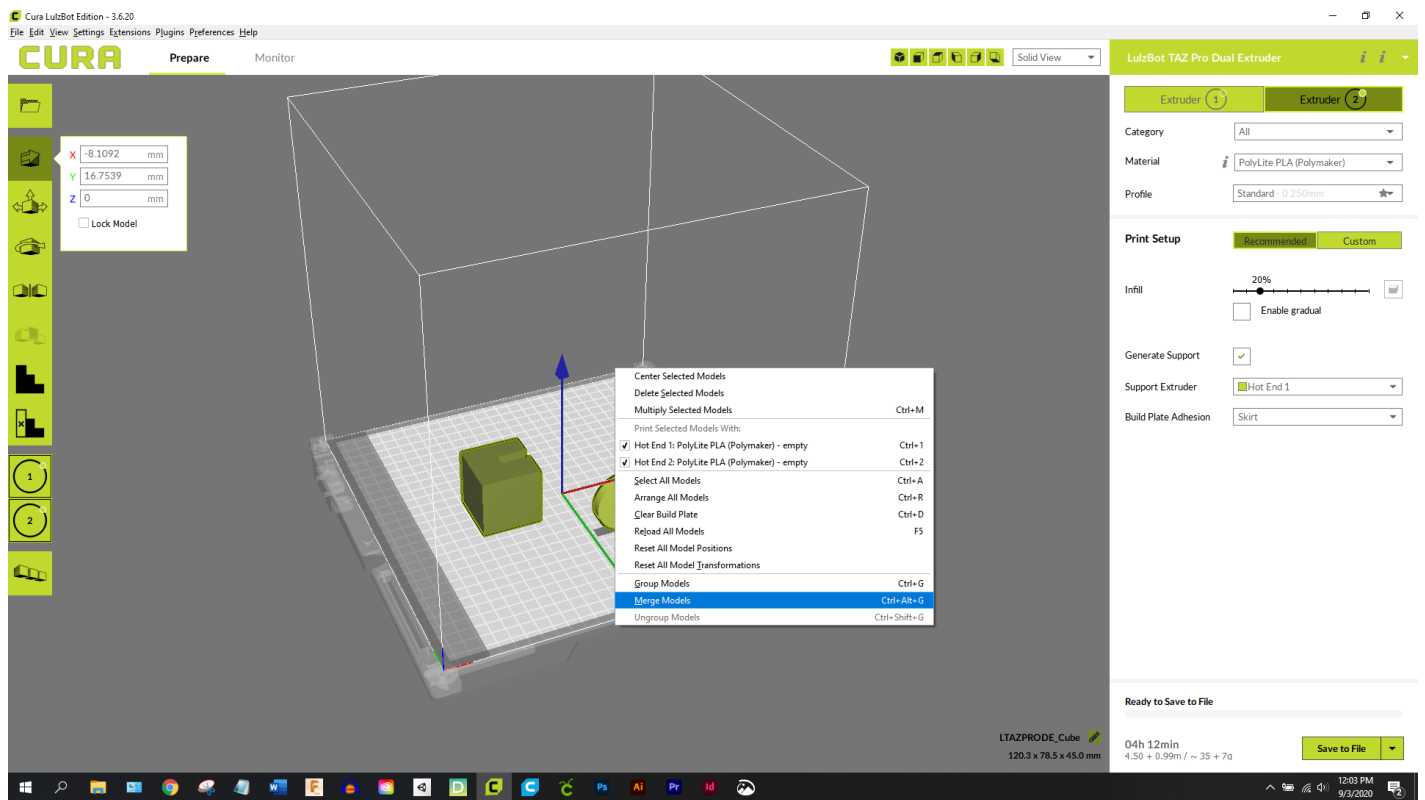
6. Repeat steps #3-5 using the second model, but make sure this time you select Hot End/Extruder 2.

The image shows a screenshot of the LulzBot TAZ Pro Dual Extruder software interface. On the left is a vertical sidebar with various icons. The icon for 'Extruder 2' is circled in black. A large black arrow points from this icon to the 'Extruder 2' tab in the settings panel on the right. The settings panel is titled 'LulzBot TAZ Pro Dual Extruder' and shows two tabs: 'Extruder 1' and 'Extruder 2'. The 'Extruder 2' tab is selected and circled in black. Below the tabs, the settings for 'Extruder 2' are displayed: Category (All), Material (PolyLite PLA (Polymaker)), Profile (Standard - 0.250mm), Print Setup (Recommended/Custom), Infill (20%), Enable gradual (unchecked), Generate Support (checked), Support Extruder (Hot End 1), and Build Plate Adhesion (Skirt). At the bottom, the interface shows 'Ready to Save to File', a progress bar, and a 'Save to File' button. The estimated print time is 06h 31min, 10.23m / ~ 81g.

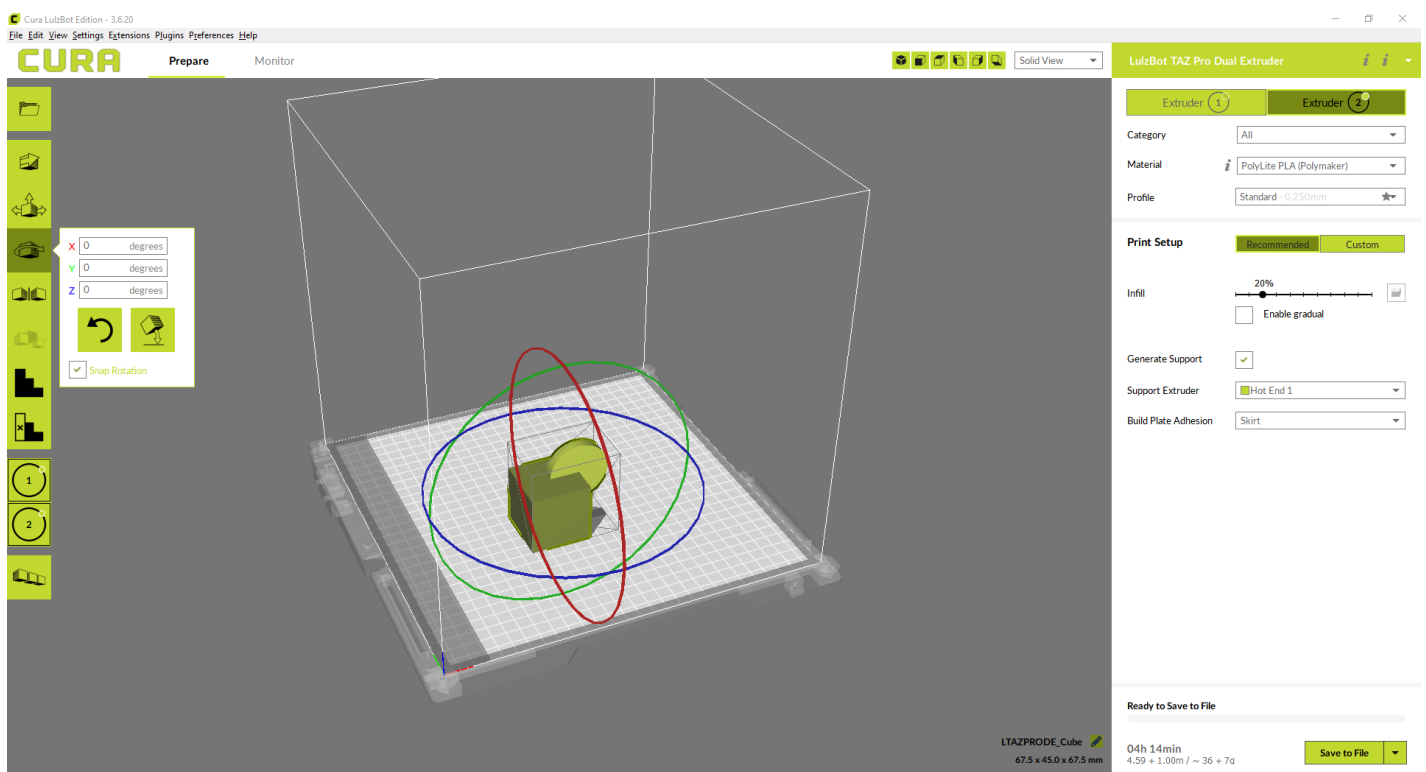
7. After you have assigned your Hot Ends/Extruders to the correct components, press “Ctrl & A” on your keyboard to select all. (Command & A on a Mac)



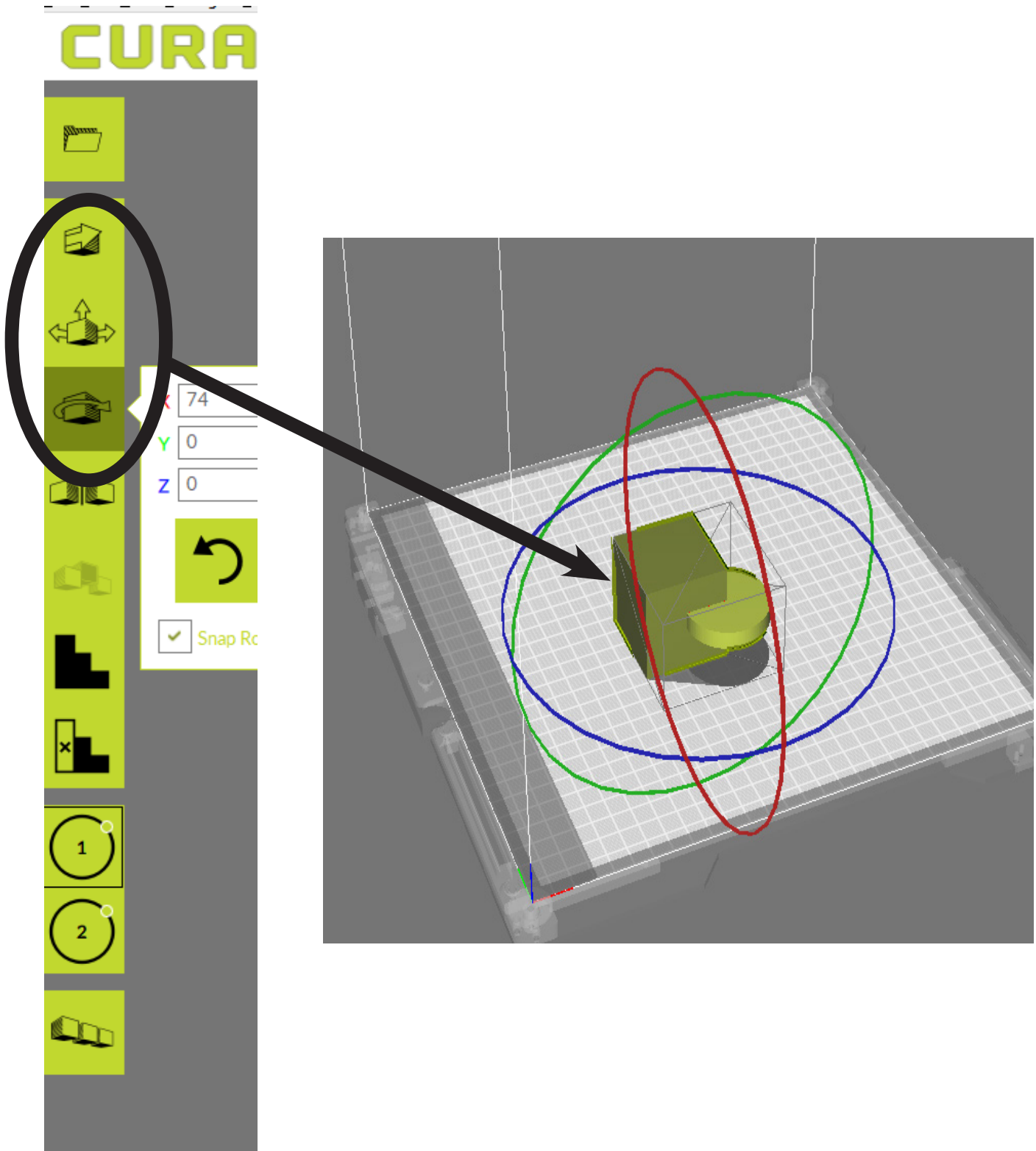
8. While all models are selected, right-click on the screen. This will bring up pop-up menu.



9. Click on the selection “Merge Models”. This will merge your models exactly how they were oriented when you exported them from your CAD program. NOTE: If they are not merged correctly, go back to your CAD file and trouble shoot.



10. After your models have been properly merged, use the icons on the left toolbar to adjust your model (Scale, Rotate, Move, etc.)



11. In the drop-down menu labeled “Profile” you can adjust the resolution of your print. Keep in mind the trade-offs: High Detail = longer print time, but better surface finish; High Speed = Less resolution but a faster print time

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
 Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

12. If needed, adjust your infill. The default setting is 20%, which is fine for most prints. However, again there is a trade-off: Higher Infill Density = stronger print, but slower print time; Lower Infill Density = Weaker print (easier to damage) but faster print time

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
Enable gradual

Generate Support

Support Extruder Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

13. If your model has over-hangs, make sure to click the “Generate Support” check-box. (NOTE: this will give you the option to use either Hot End/Extruder to print the support structure.)

The screenshot displays the configuration interface for a LulzBot TAZ Pro Dual Extruder. At the top, there are tabs for 'Extruder 1' and 'Extruder 2'. Below these are settings for 'Category' (All), 'Material' (PolyLite PLA (Polymaker)), and 'Profile' (Standard - 0.250mm). The 'Print Setup' section includes 'Recommended' and 'Custom' tabs, an 'Infill' slider set to 20% with an 'Enable gradual' checkbox, and a 'Generate Support' checkbox which is checked. Below this is a 'Support Extruder' dropdown menu set to 'Hot End 1' and a 'Build Plate Adhesion' dropdown menu set to 'Skirt'. At the bottom, the interface shows 'Ready to Save to File', a progress bar, and a 'Save to File' button. Estimated print time is 06h 13min and weight is 10.17m / ~ 81g.

14. In the drop-down menu labeled “Build Plate Adhesion”, select the style of adhesion you want. (NOTE: It is NOT recommended to select “NONE”, as this increases the likelihood of a failed print due to the model “peeling” away from the build platform during the print)

LulzBot TAZ Pro Dual Extruder

Extruder 1 Extruder 2

Category All

Material PolyLite PLA (Polymaker)

Profile Standard - 0.250mm

Print Setup Recommended Custom

Infill 20%
 Enable gradual

Generate Support

Support Material Hot End 1

Build Plate Adhesion Skirt

Ready to Save to File

06h 13min
10.17m / ~ 81g

Save to File

15. If you are finished making all adjustments to both the model and print settings, insert a USB drive into your computer and click the “Save to Removable Drive” button in the lower right-hand part of the screen. (NOTE: You may need to click the arrow to change the selection from “Save to File” to “Save to Removable Drive”)

The screenshot shows the LulzBot TAZ Pro Dual Extruder software interface. At the top, there is a green header with the text "LulzBot TAZ Pro Dual Extruder" and two information icons. Below this, there are two tabs for "Extruder 1" and "Extruder 2". The "Extruder 1" tab is selected. Underneath, there are three dropdown menus: "Category" set to "All", "Material" set to "PolyLite PLA (Polymaker)", and "Profile" set to "Standard - 0.250mm". Below these is a "Print Setup" section with two tabs: "Recommended" and "Custom". The "Recommended" tab is selected. Under "Print Setup", there is an "Infill" slider set to 20%, an "Enable gradual" checkbox, a "Generate Support" checkbox with a checkmark, a "Support Extruder" dropdown set to "Hot End 1", and a "Build Plate Adhesion" dropdown set to "Skirt".



16. Remove your USB from your computer and insert it into the Taz Pro



MACHINE OPERATION

1. After you insert your USB Drive into the printer, apply a layer of glue to the build platform using a glue stick



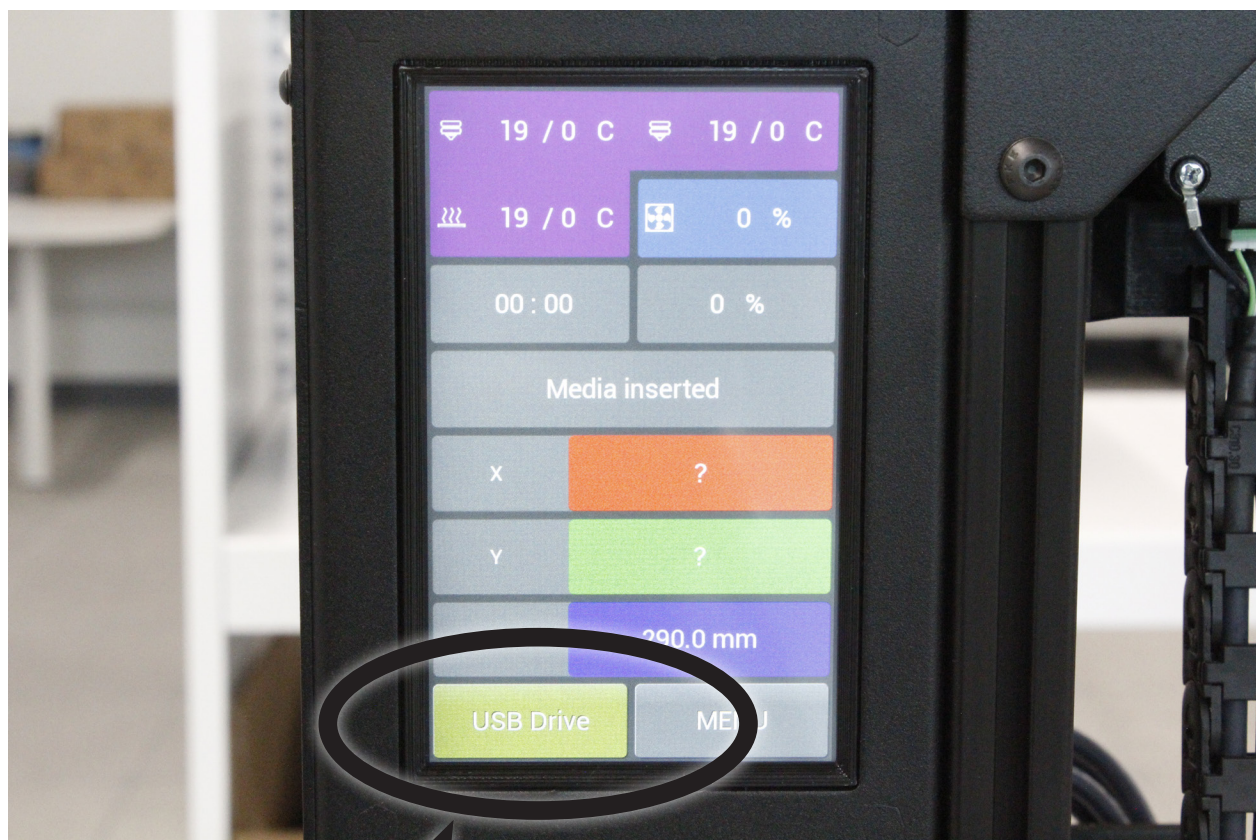
2. Turn on the printer



3. Ensure the proper filament is loaded into the printer. (The left extruder is #1 and the right extruder is #2) (NOTE: If you need to change the filament, please ask one of the Shop Technicians for assistance.)



4. On the printer touch-screen, press the “USB Drive” button located in the lower left corner of the screen



5. Locate the file you wish to print and tap the title of the print on the touch-screen. (Your selection will be highlighted)

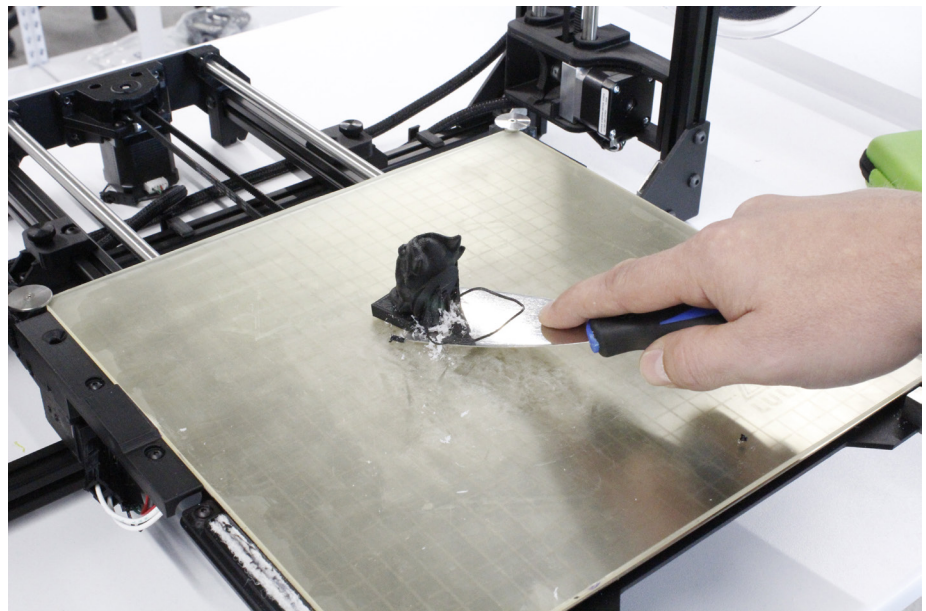
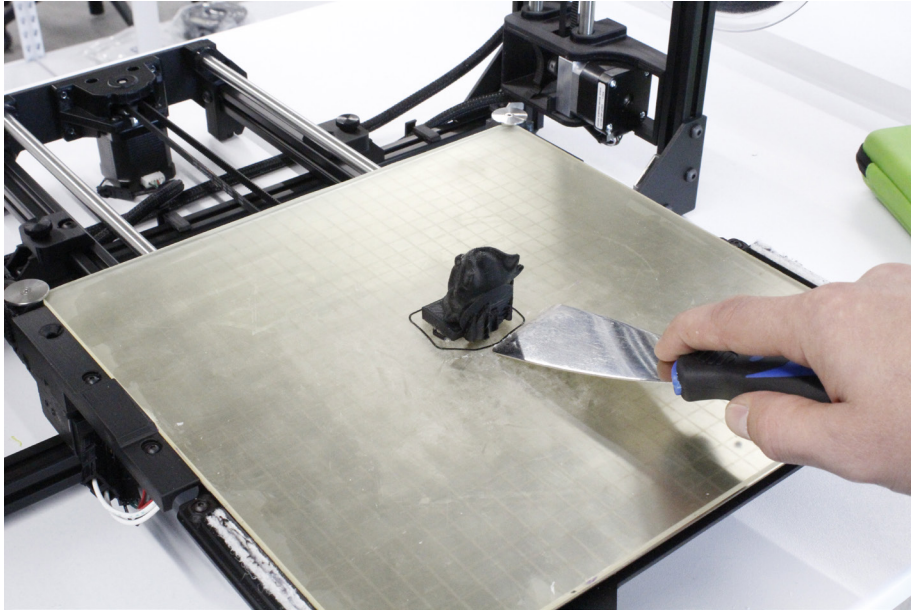


6. Press the “Print” button on the touch-screen. (NOTE: It is a very good idea to monitor the print for about 15-20 minutes to ensure the print is successful, as most prints fail during this time)

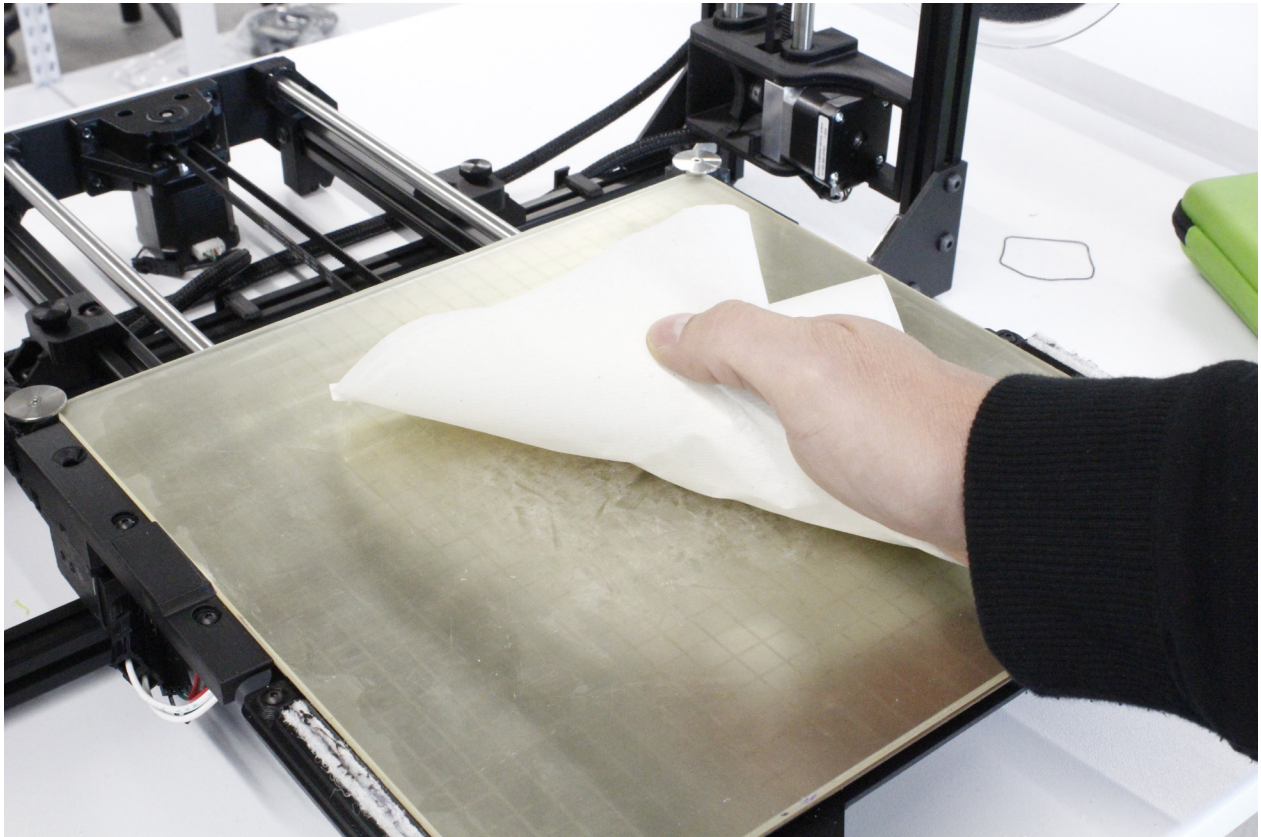


NOTE: Both the extruder and the build platform may be very hot immediately after the print finishes. These components can burn you if you touch them. Therefore, wait a short period of time before removing your print.

1. Using the scraper, gently, yet firmly, remove the print from the build platform. (NOTE: The scraper is SHARP!!! It can easily cut you. ALWAYS scrape AWAY from any body parts)



2. Using a paper towel with a little Isopropyl Alcohol, wipe off any excess filament. (You may need to use the scraper for filament that is difficult to remove with a paper towel, be careful!)



3. On the printer's touch-screen, press the "Menu" button located in the lower right corner



4. Press the “Auto Home” button in the top left corner of the touch-screen

