

# ULTIMAKER S5 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 PROCEEDURES.....6

OPERATIONS GUIDE.....7

    SOFTWARE INTERFACE .....8

    SOFTWARE OPERATION (SINGLE EXTRUDER) .....10

    SOFTWARE OPERATION (DUAL EXTRUDER) .....25

    MACHINE OPERATION .....44

    AFTER PRINTING.....51

# OVERVIEW

The purpose of this document is to provide standard operating procedures for the use of the Ultimaker S5 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.

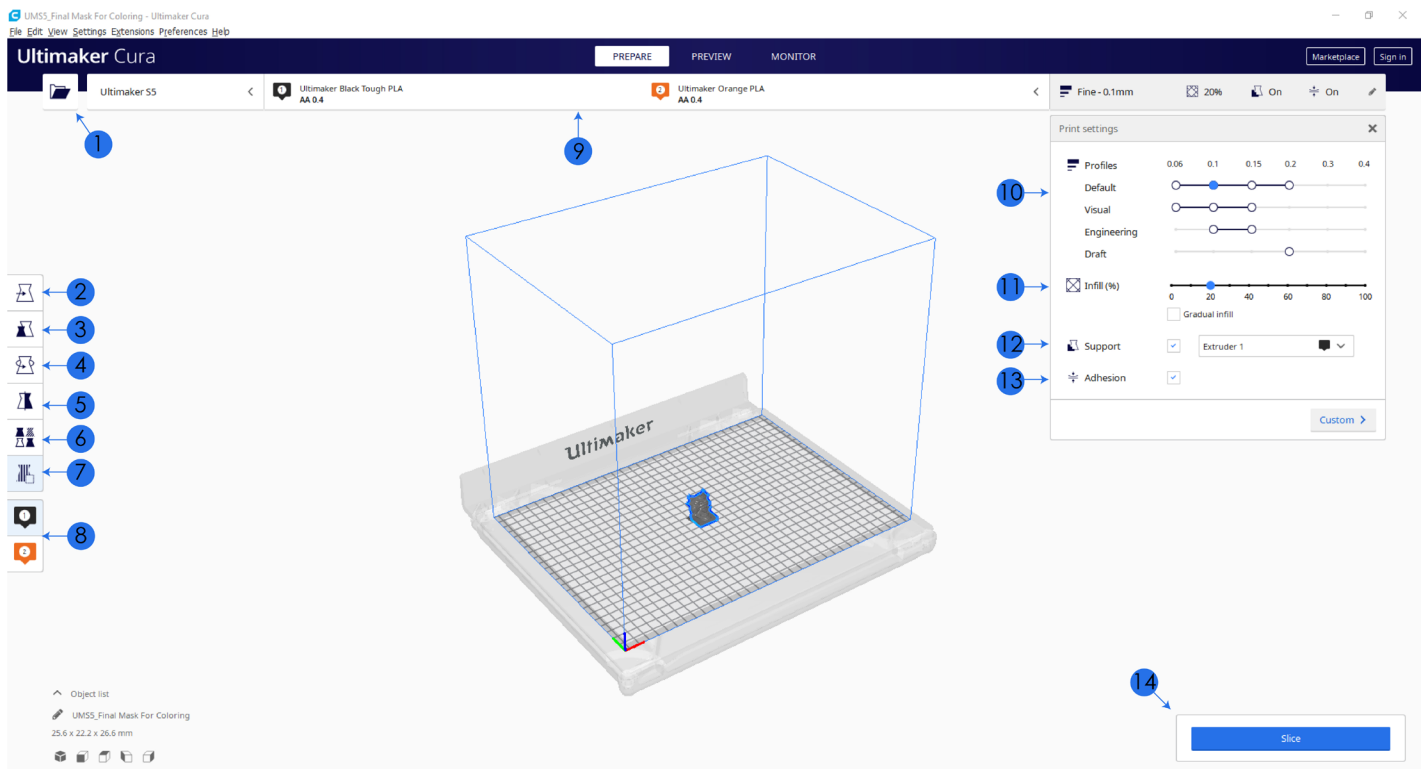
# SPILL/CLEAN UP PROCEDURES

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

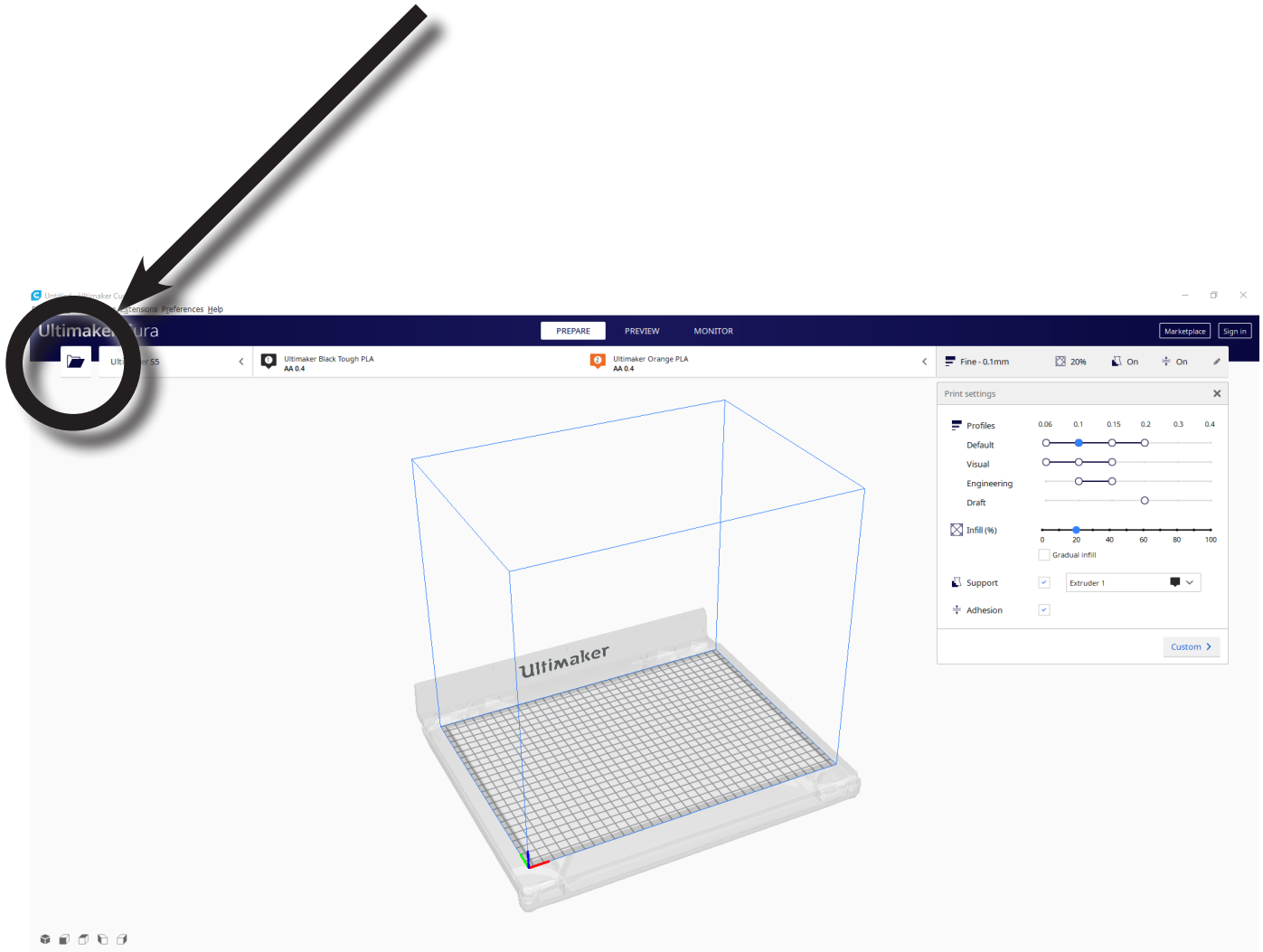


- 1. Open File:** Import the file to be printed.
- 2. Move Model:** Move the model
- 3. Scale Model:** Adjust the size of the model. (NOTE: Cura uses metric, not imperial)
- 4. Rotate Model:** Rotate the model via X, Y, Z axis
- 5. Mirror Model:** Creates a reverses/mirrored image of the model selected 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. Print Core Selection:** This allows you to choose which Print Core/Extruder you wish to use for your print.
- 9. Extruder Settings:** Allows you to Enable/disable extruders, assign material types, and Print Core configurations
- 10. Print Profile:** The print resolution. Keep in mind the trade-offs: a higher resolution like 0.06 = longer print time, but better surface finish; a lower resolution like 0.2 = Less resolution but a faster print time
- 11. Infill:** How solid the interior of the print is in percentage
- 12. Generate Support:** Used to create support structures for your print as well as which Extruder is used for supports.
- 13. Build Plate Adhesion:** Used to generate an adhesion that will help prevent warping and peeling of the print. The default adhesion type is a "Brim". Brims add a single layer flat area around the base of the model.
- 14. Slice/Save to File/Save to Removable Drive:** Allows you to Slice (prepare the model for the printer) and then save your prepared file to either your computer or a USB drive.

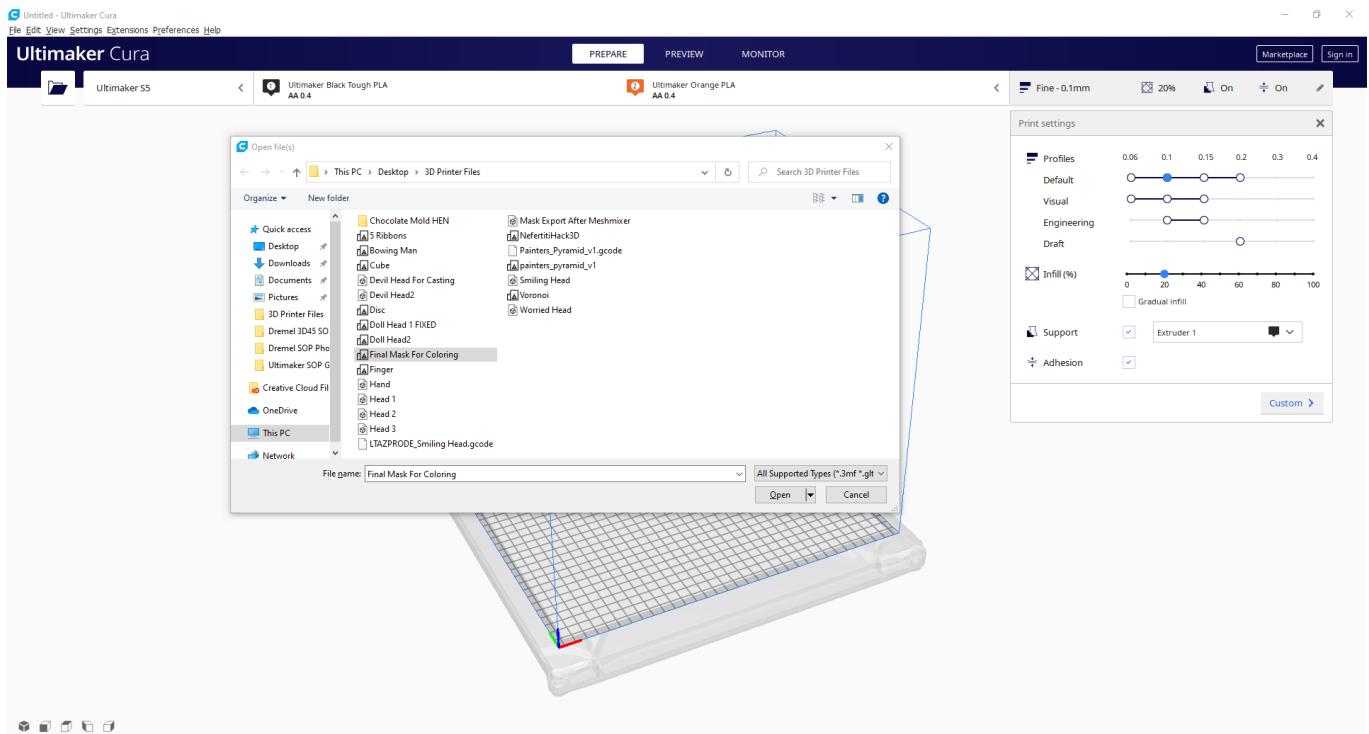
## SOFTWARE OPERATION (SINGLE EXTRUDER)

*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 18.*

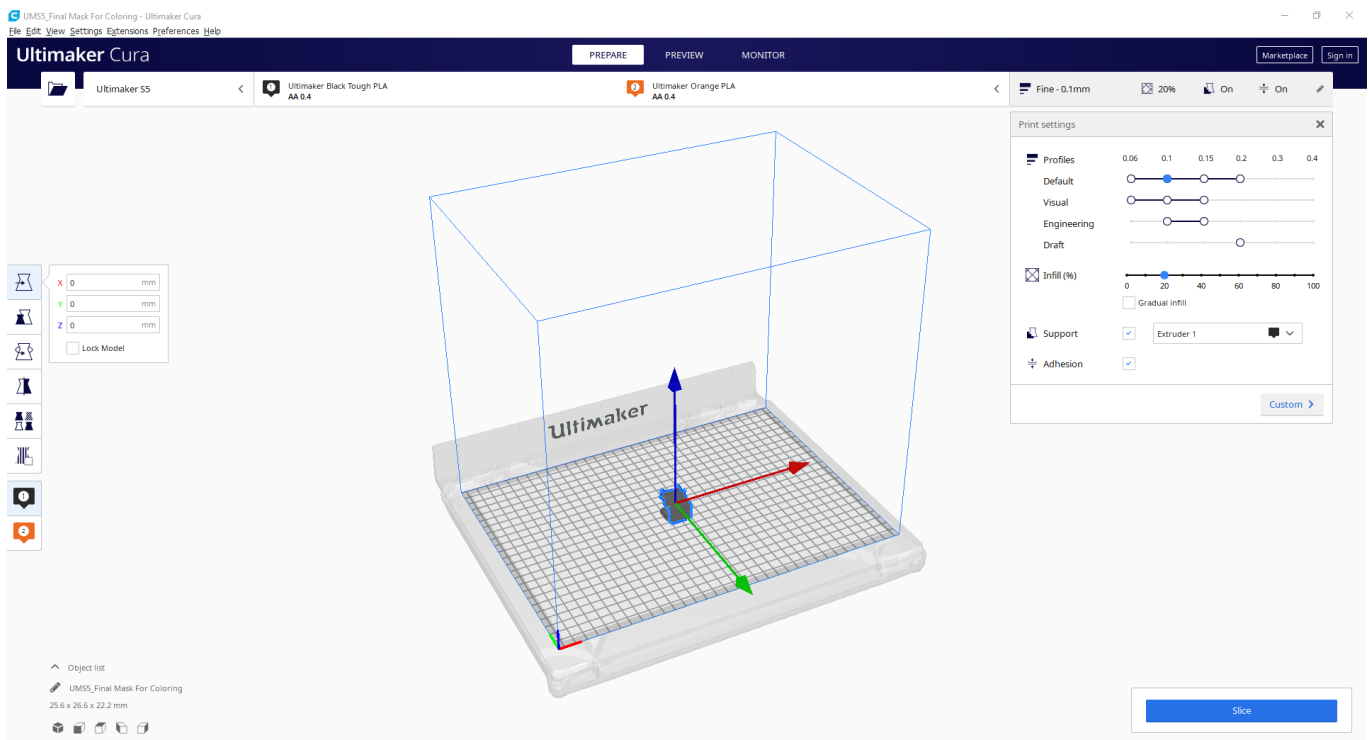
1. Open Ultimaker Cura and click the “Open File” icon in the left toolbar



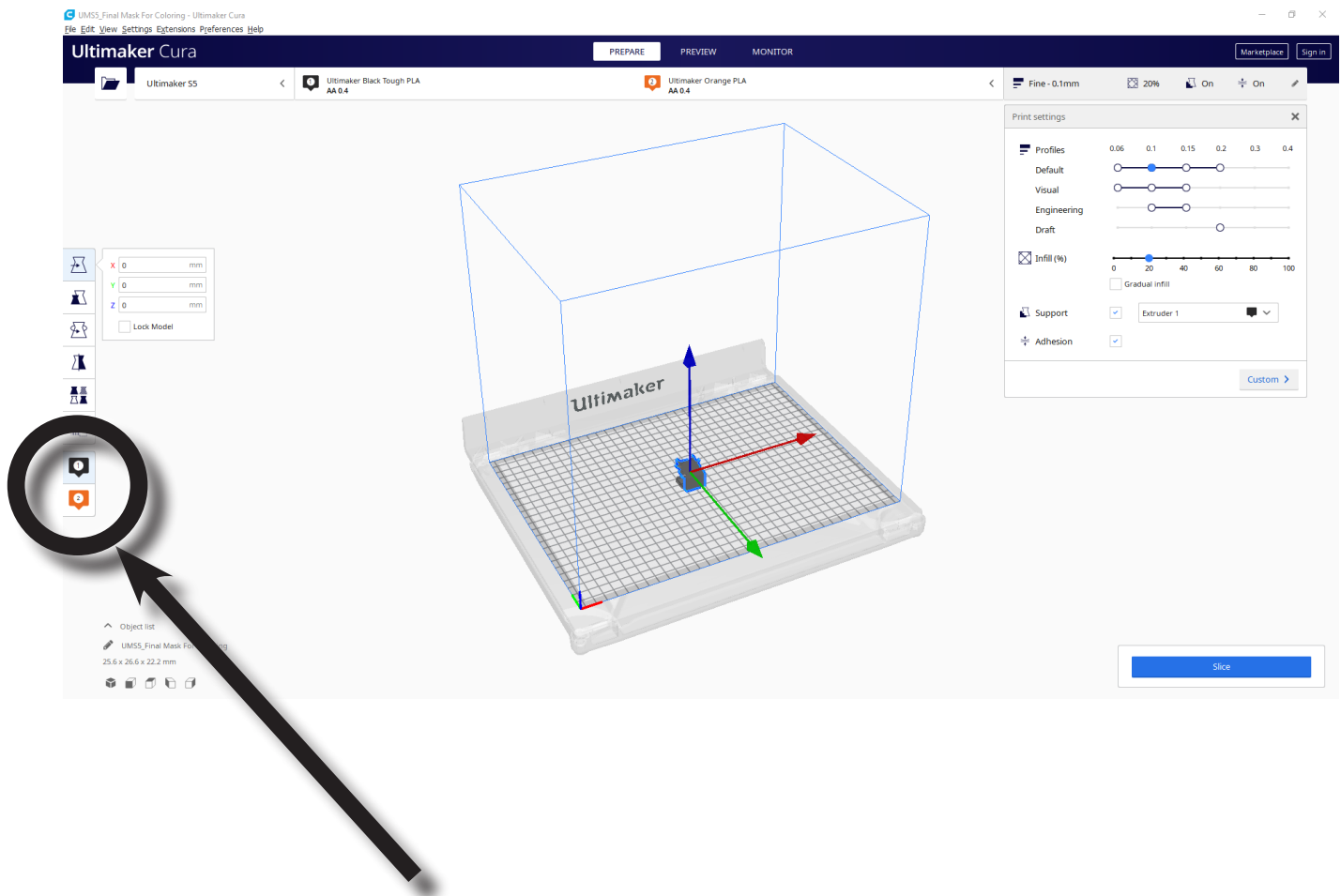
2. Locate your file and click “Open” in the dialog window



3. The model will appear in the virtual build platform. Click on it to select it

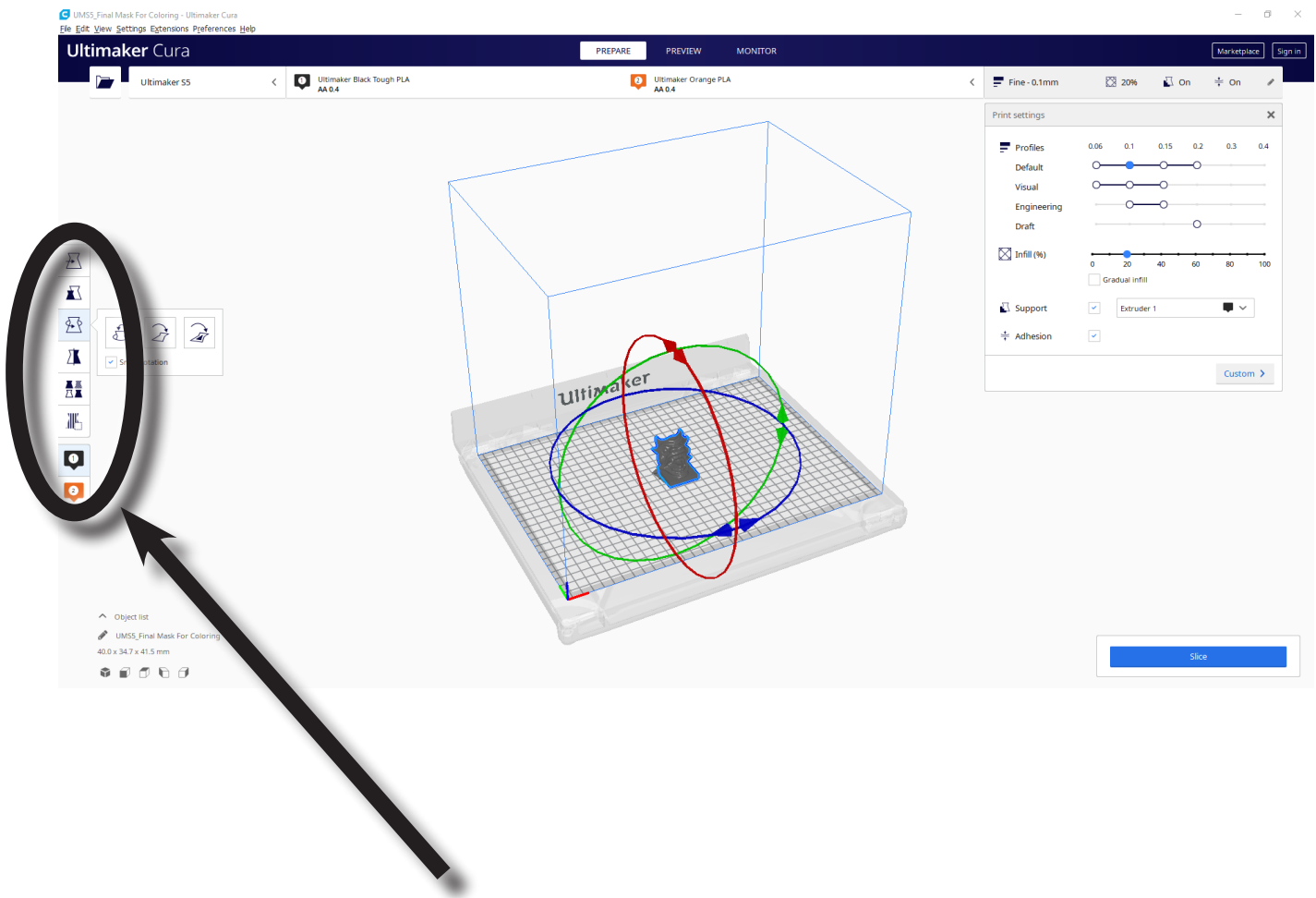


4. Choose the “Hot End” you wish to use via the left toolbar icon (1) or (2). (The icon you select will be shaded)

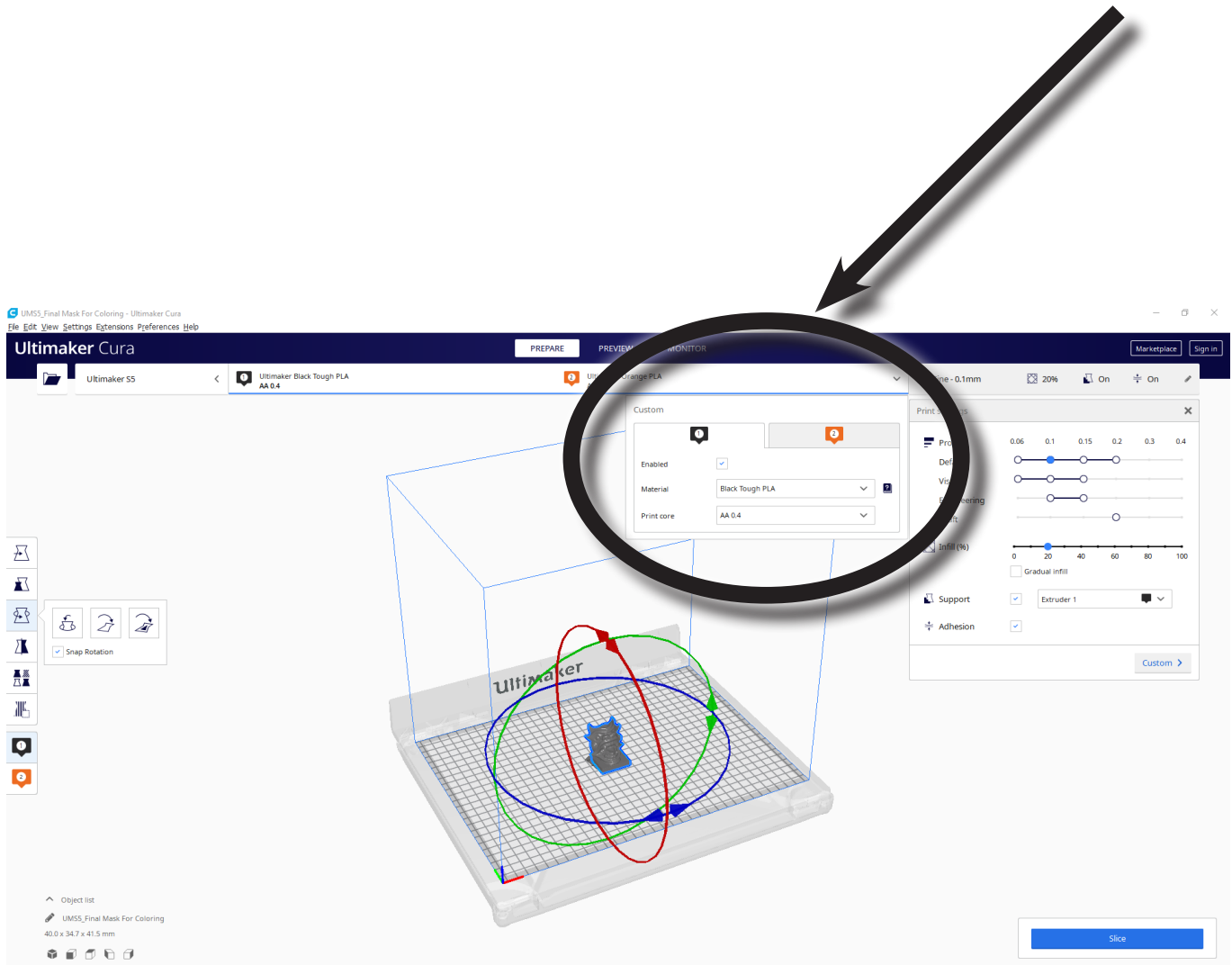




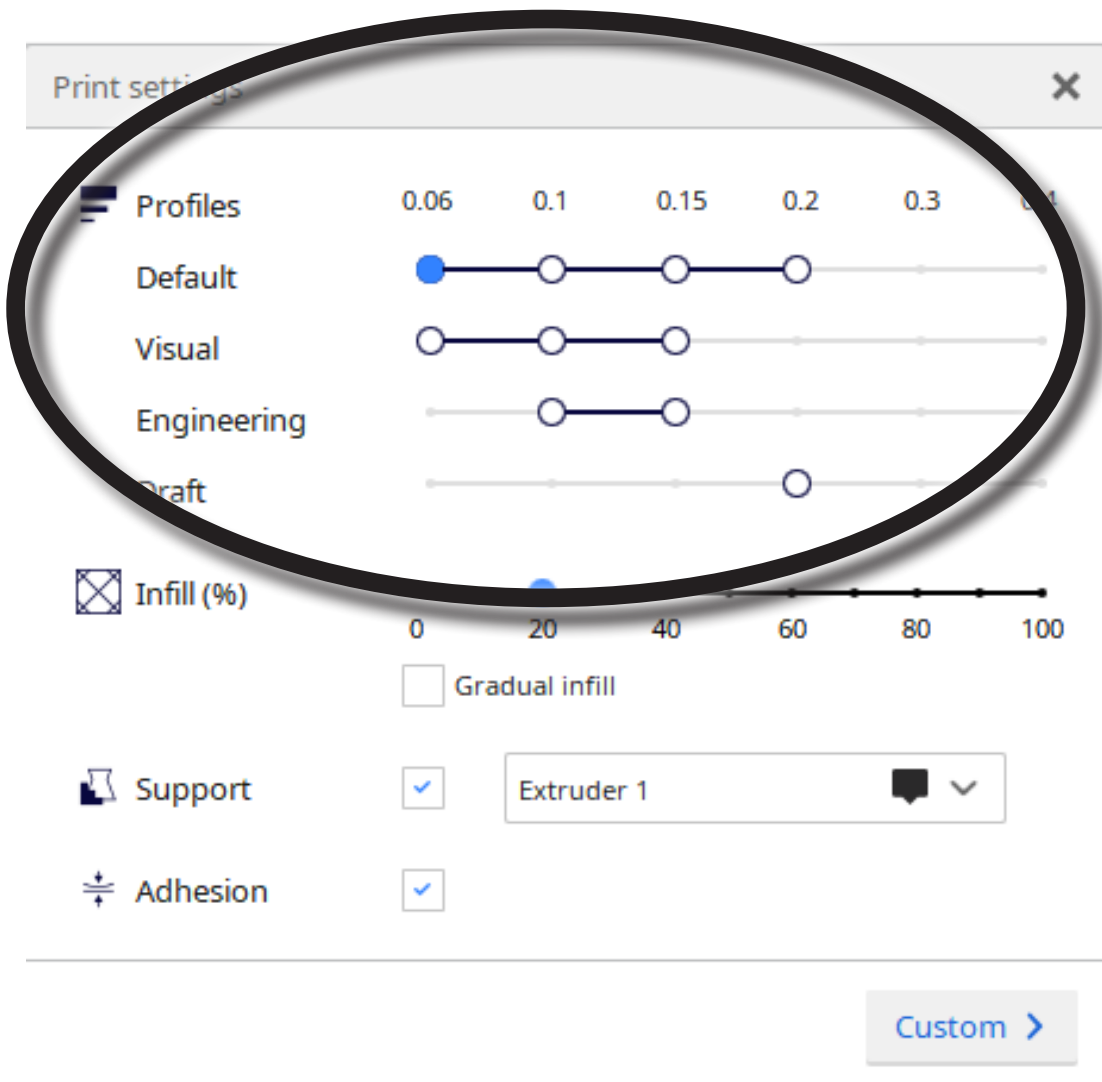
5. If needed, use the icons on the left toolbar to adjust your model (Scale, Rotate, Move, etc.)



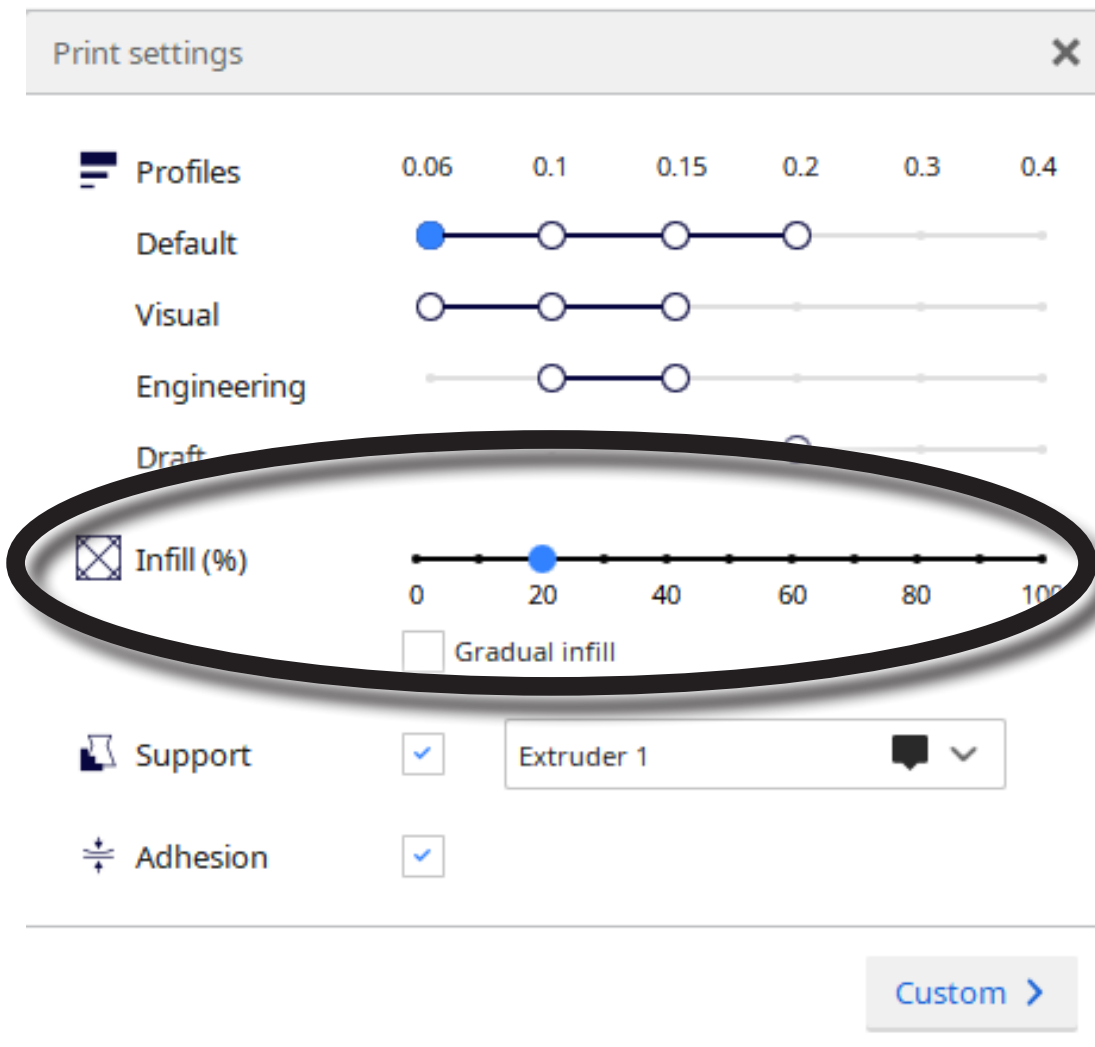
6. In the top toolbar, ensure the settings for Hot End #1 is: Material = PLA (or "Black Tough PLA"), Print Core = AA 0.4



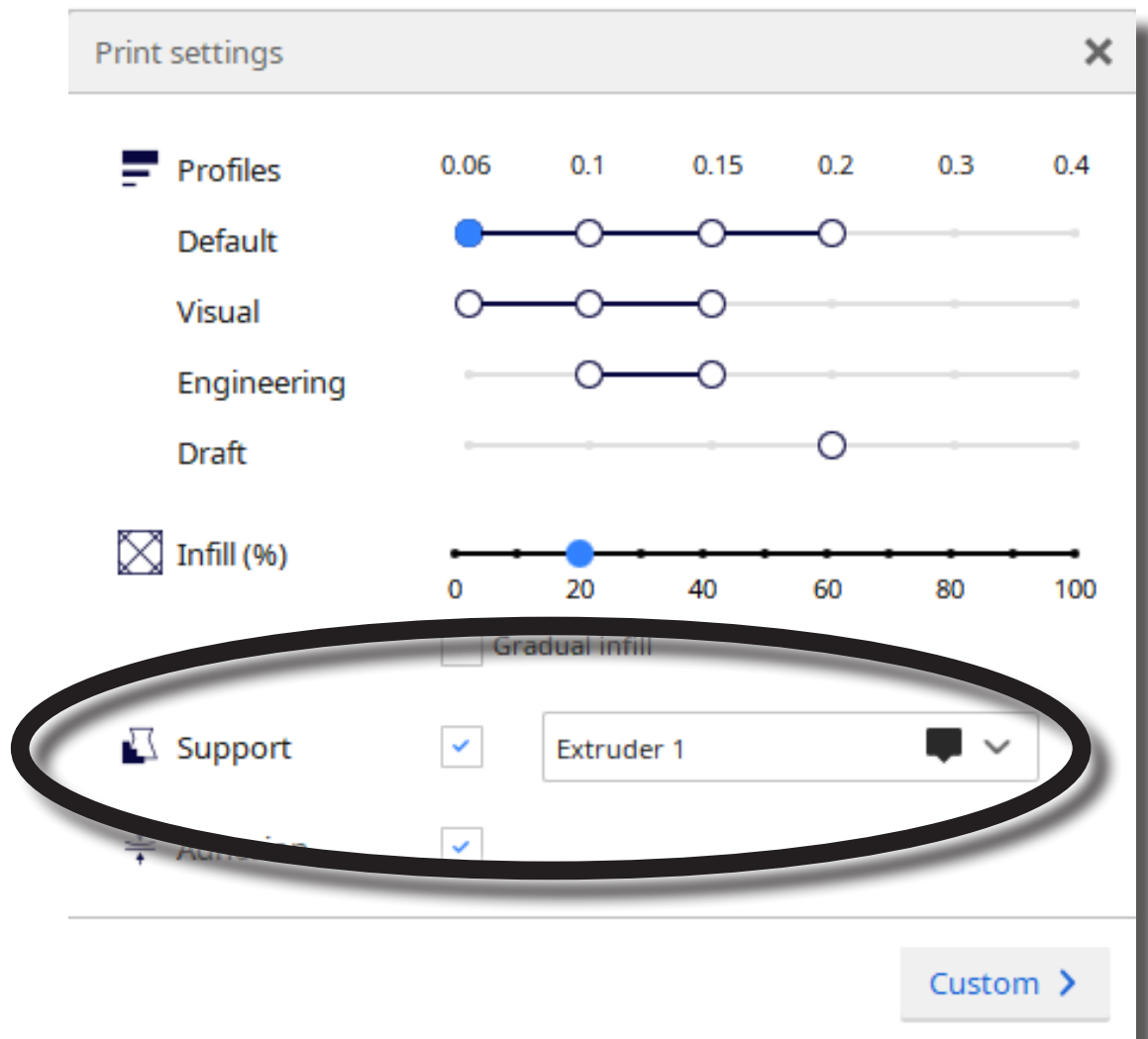
7. In the right toolbar, select your Profile. This is the print resolution. Keep in mind the trade-offs: a higher resolution like 0.06 = longer print time, but better surface finish; a lower resolution like 0.2 = Less resolution but a faster print time



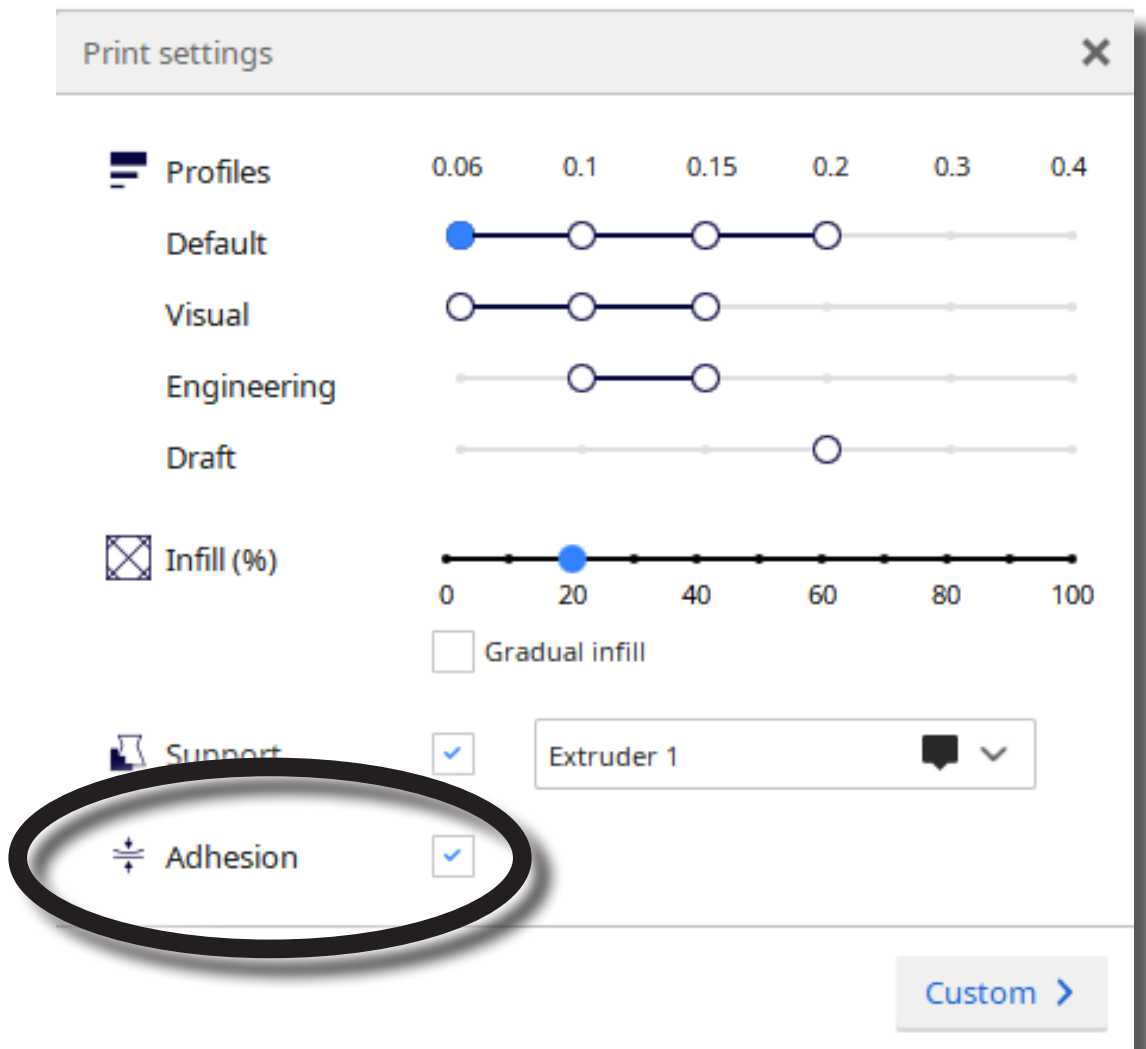
8. If needed, adjust your infill. The default setting is 20%, which is fine for most prints. However, again there is a trade-off: Higher Infill Density = stronger print, but slower print time; Lower Infill Density = Weaker print (easier to damage) but faster print time



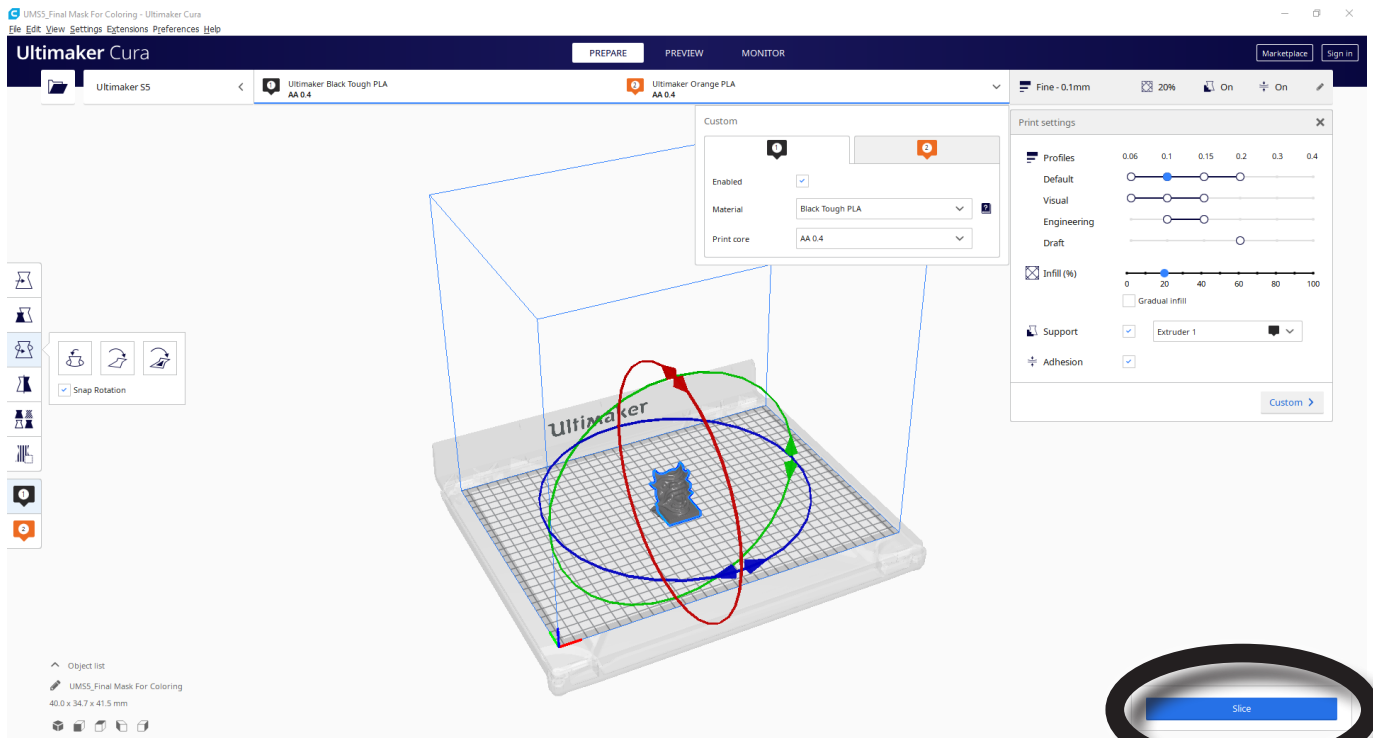
9. If your model has over-hangs, make sure to click the “Support” check-box. You can view the support structure by first slicing the model then clicking the “Preview” button at the top of the screen. Also, be sure to double check that the correct extruder is, in fact, printing the support structure.



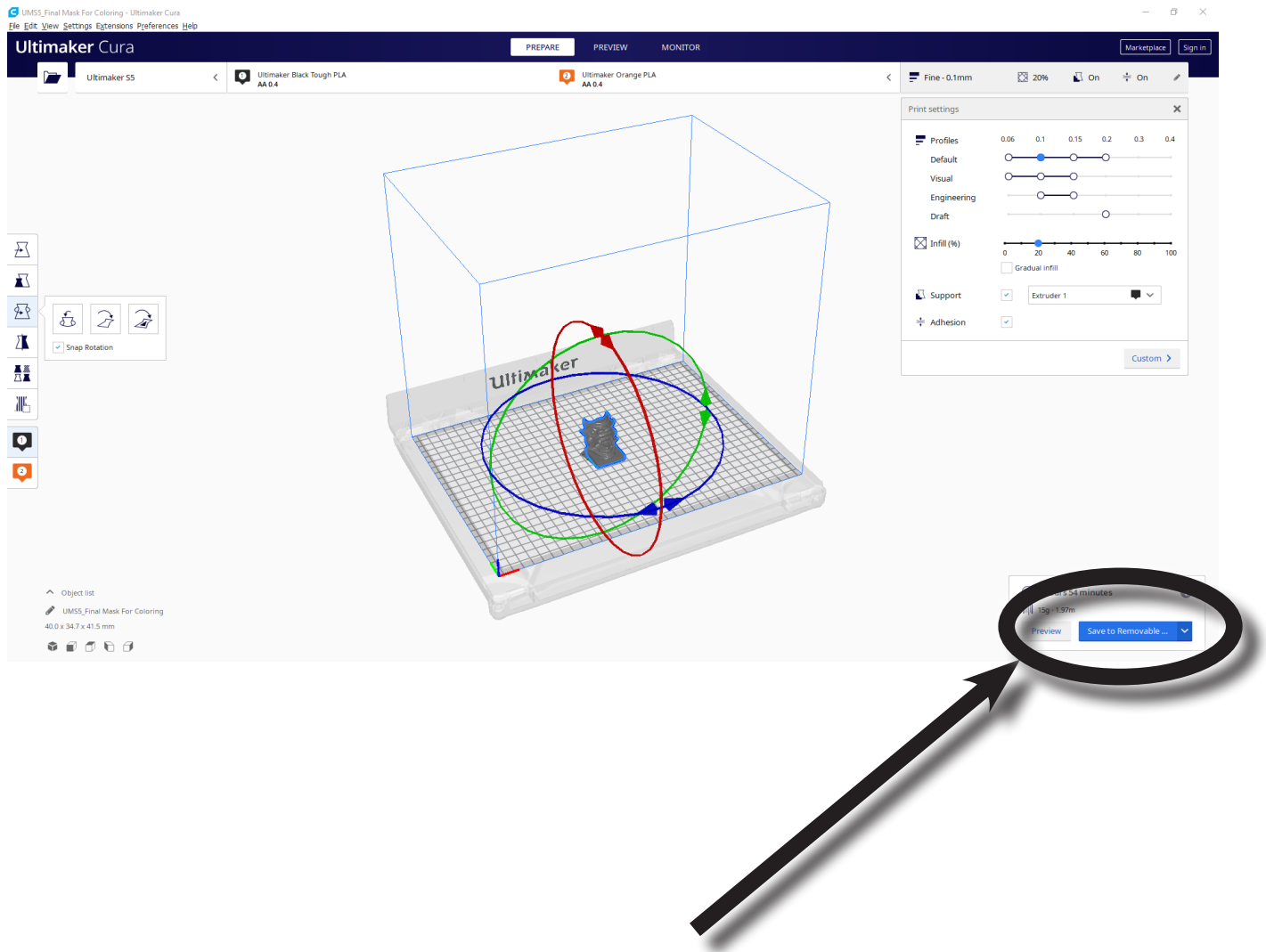
10. In the right toolbar, you can add an “Adhesion” to your print. The default adhesion type is a skirt, which places a single-layer-thick, flat area around your object, which resists the pulling forces as the print cools. (NOTE: It is NOT recommended to select “NONE”, as this increases the likelihood of a failed print due to the model “peeling” away from the build platform during the print).



11. When you have finished adjusting the model and print settings, click the “Slice” button in the lower right part of the screen

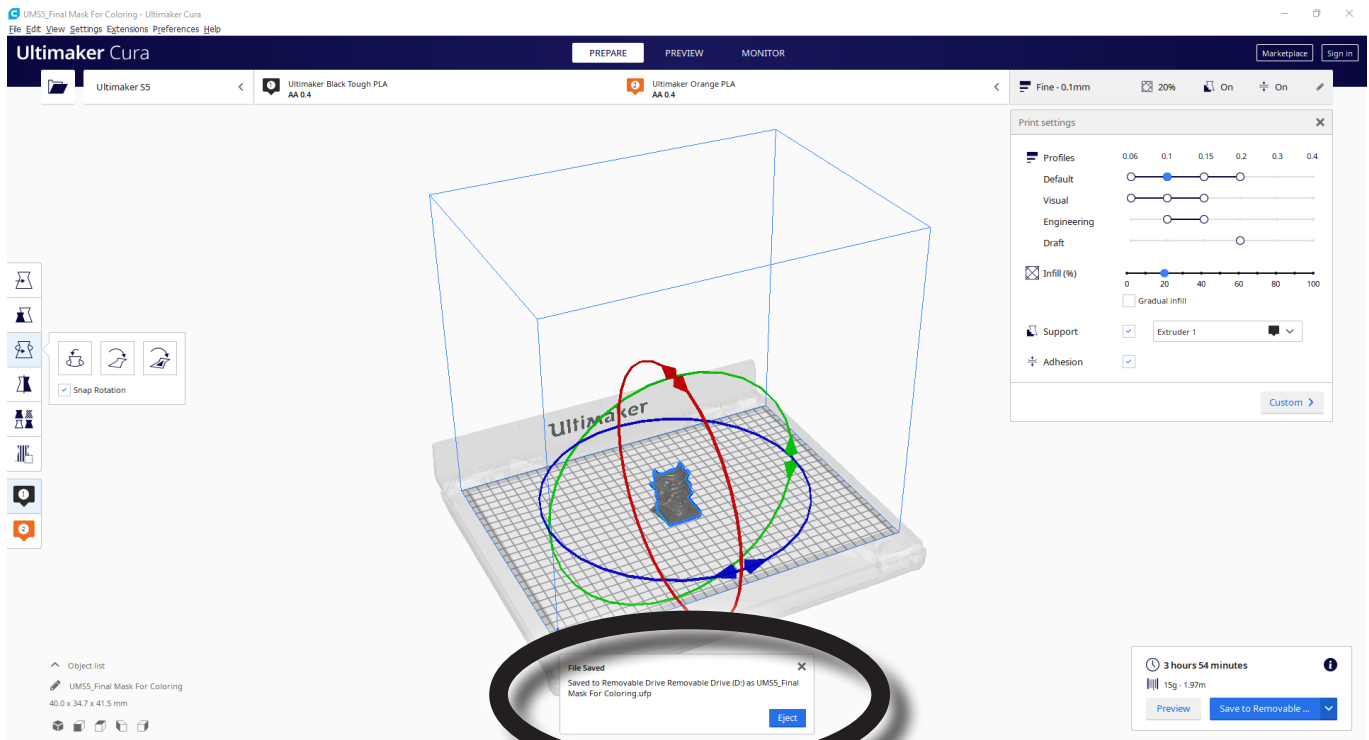


12. After Cura finishes slicing the model, insert a USB drive into your computer then click the “Save to Removable Drive” button in the lower right part of the screen





13. In the lower center part of the screen, click the “Eject” button



14. Insert your USB into the USB port on the front of the printer

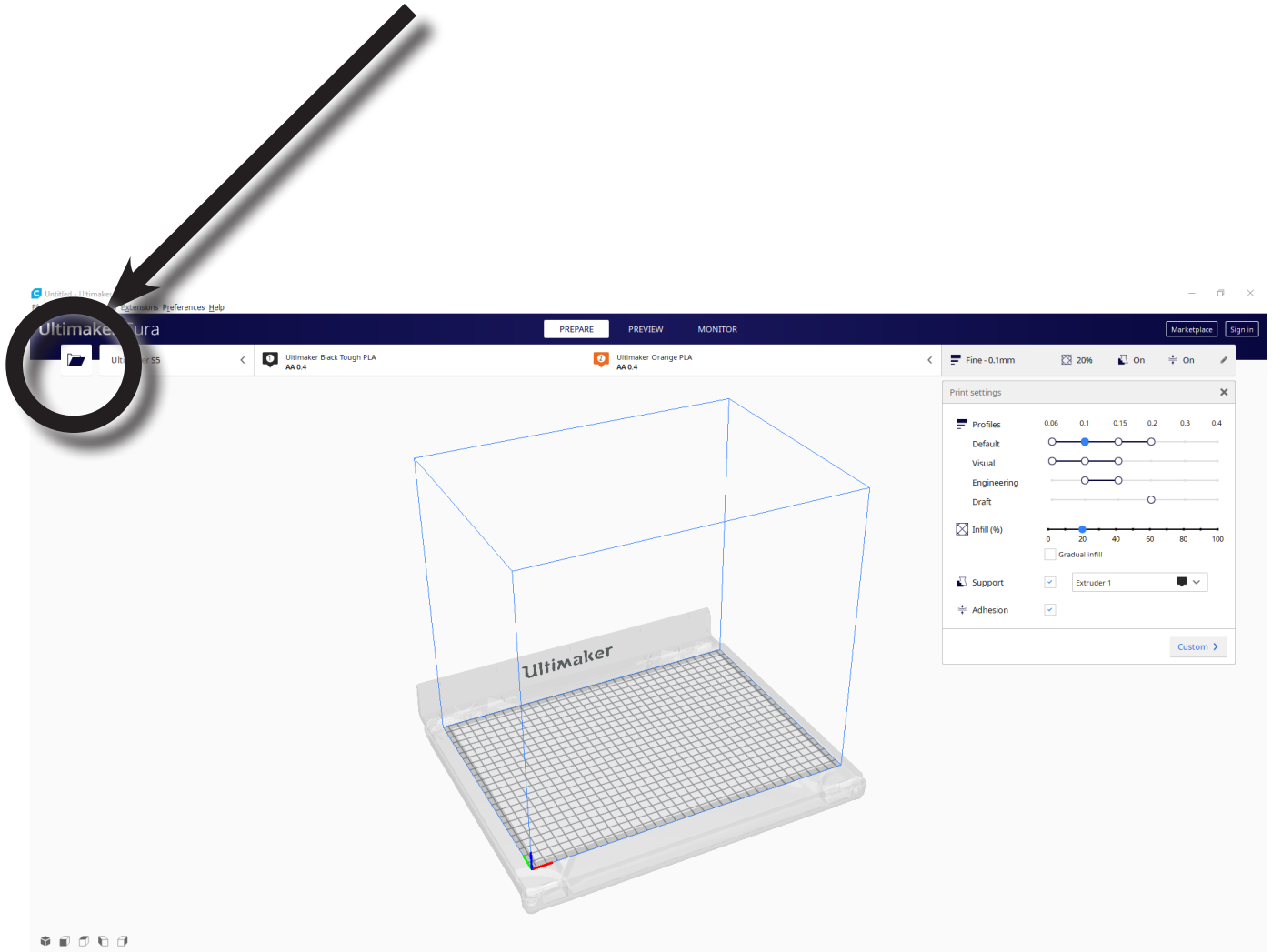


## SOFTWARE OPERATION (DUAL EXTRUDER)

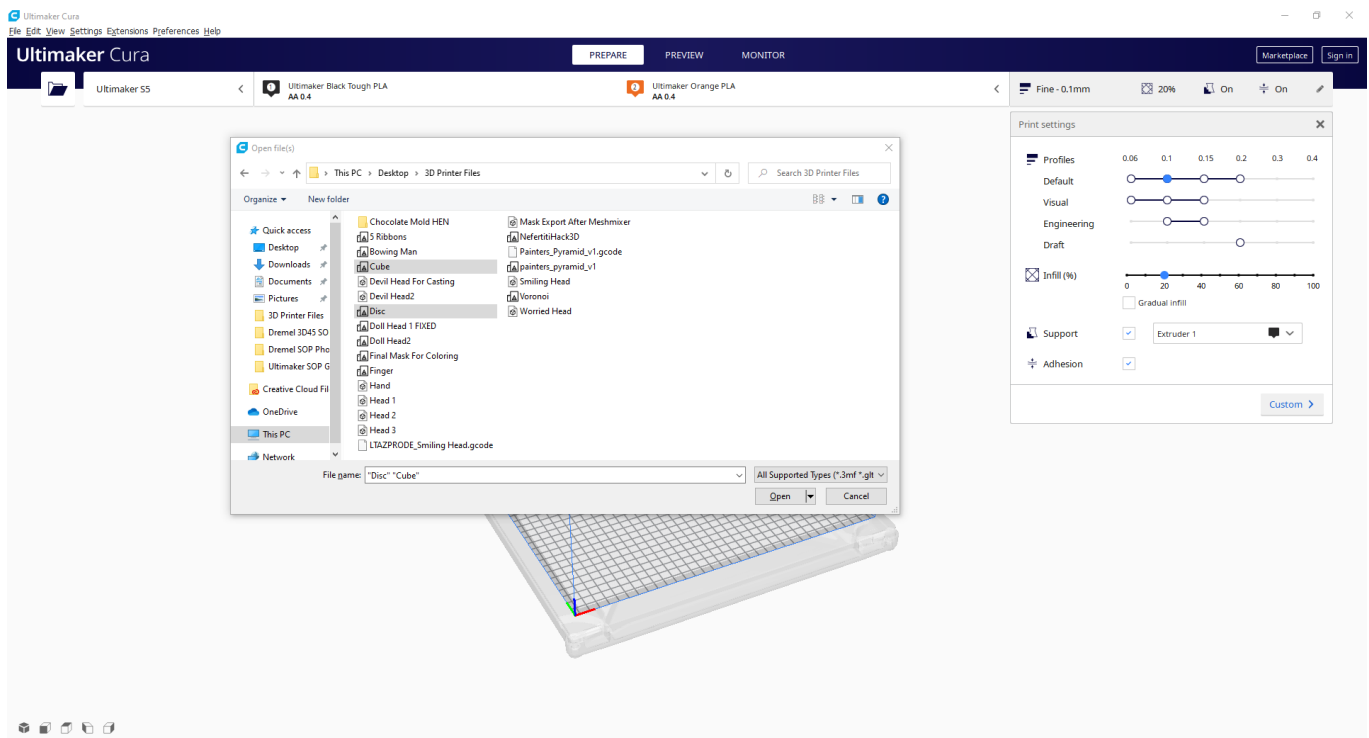
*The following steps are for using multiple colors/materials. If you are using only one color of filament or one material type, go to the section titled "SINGLE EXTRUDER" on page 03.*

*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.*

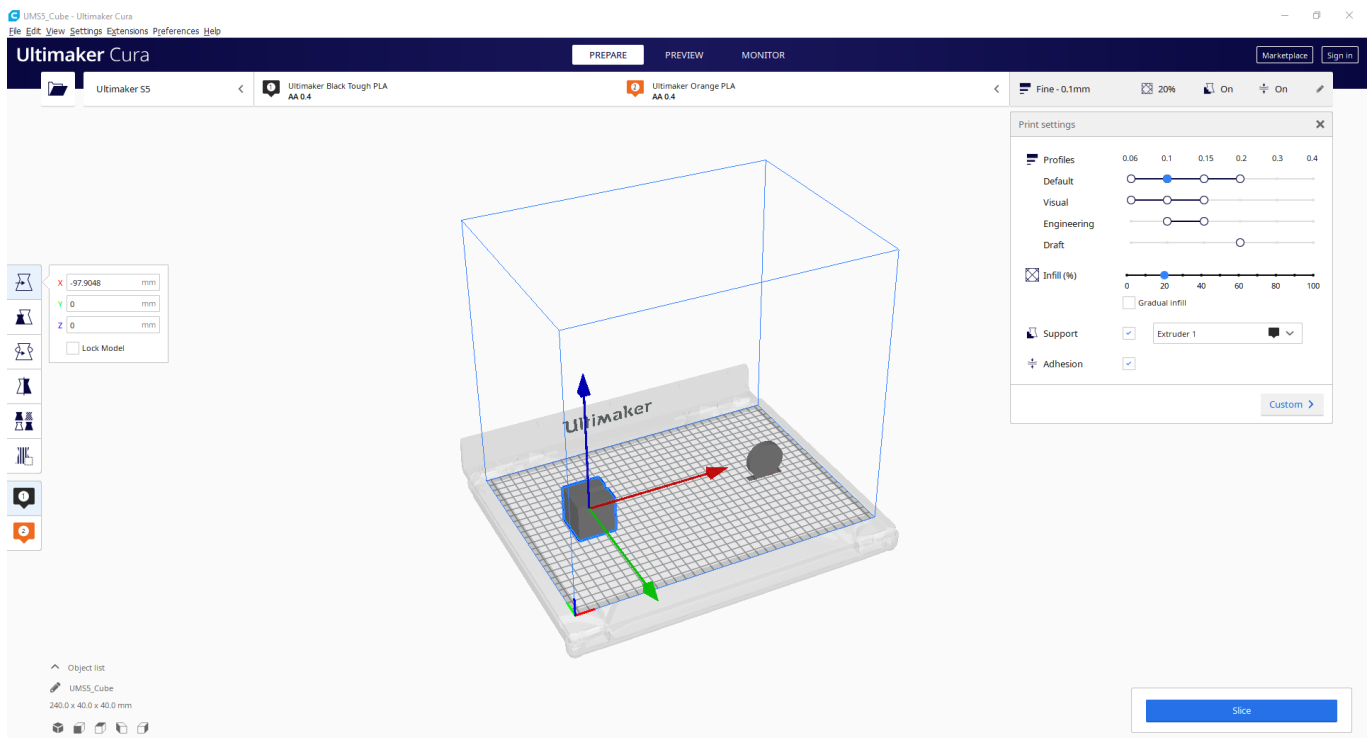
1. Click the “Open File” icon on the right toolbar.



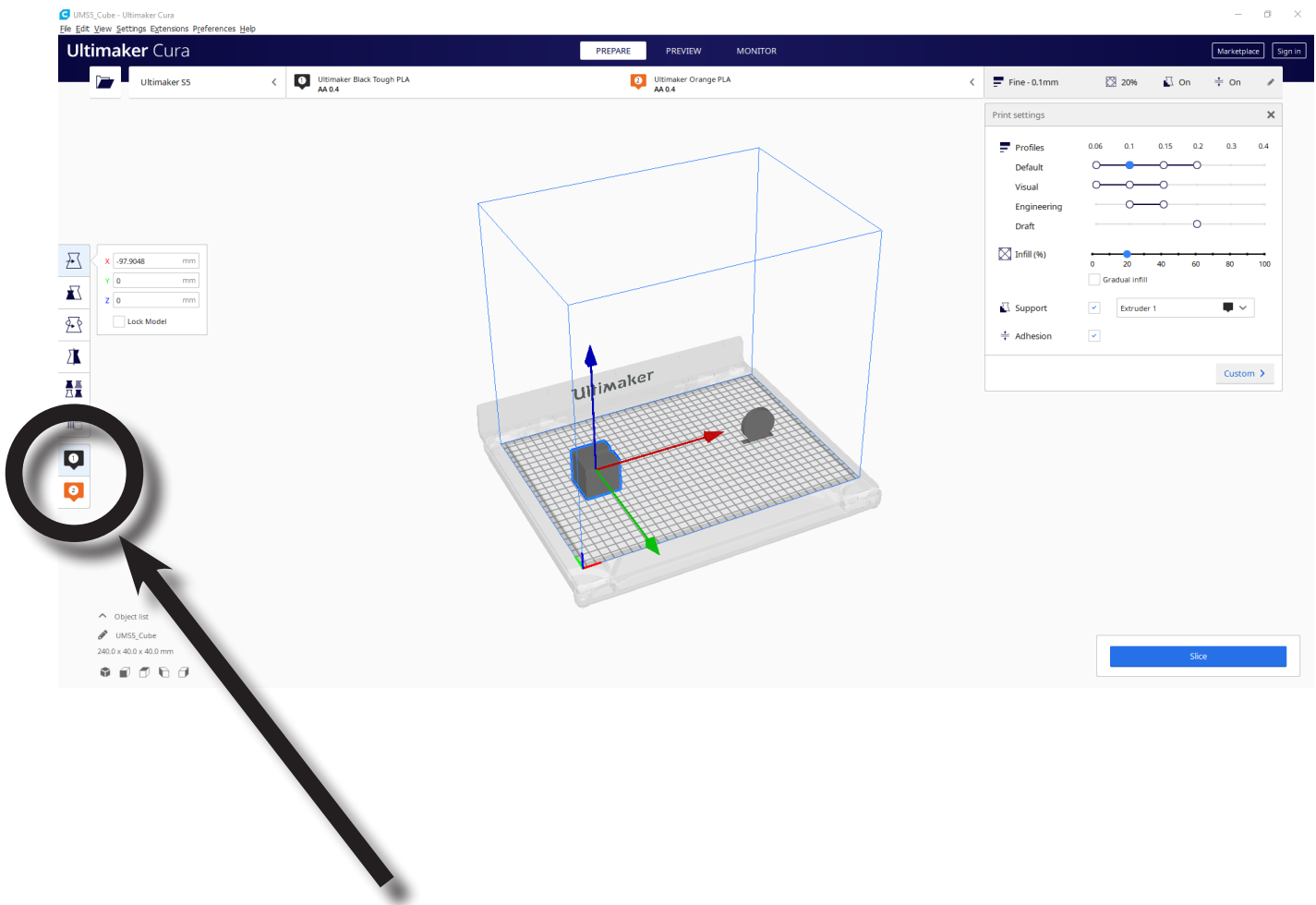
2. In the dialog window, locate your file/files and click “Open”



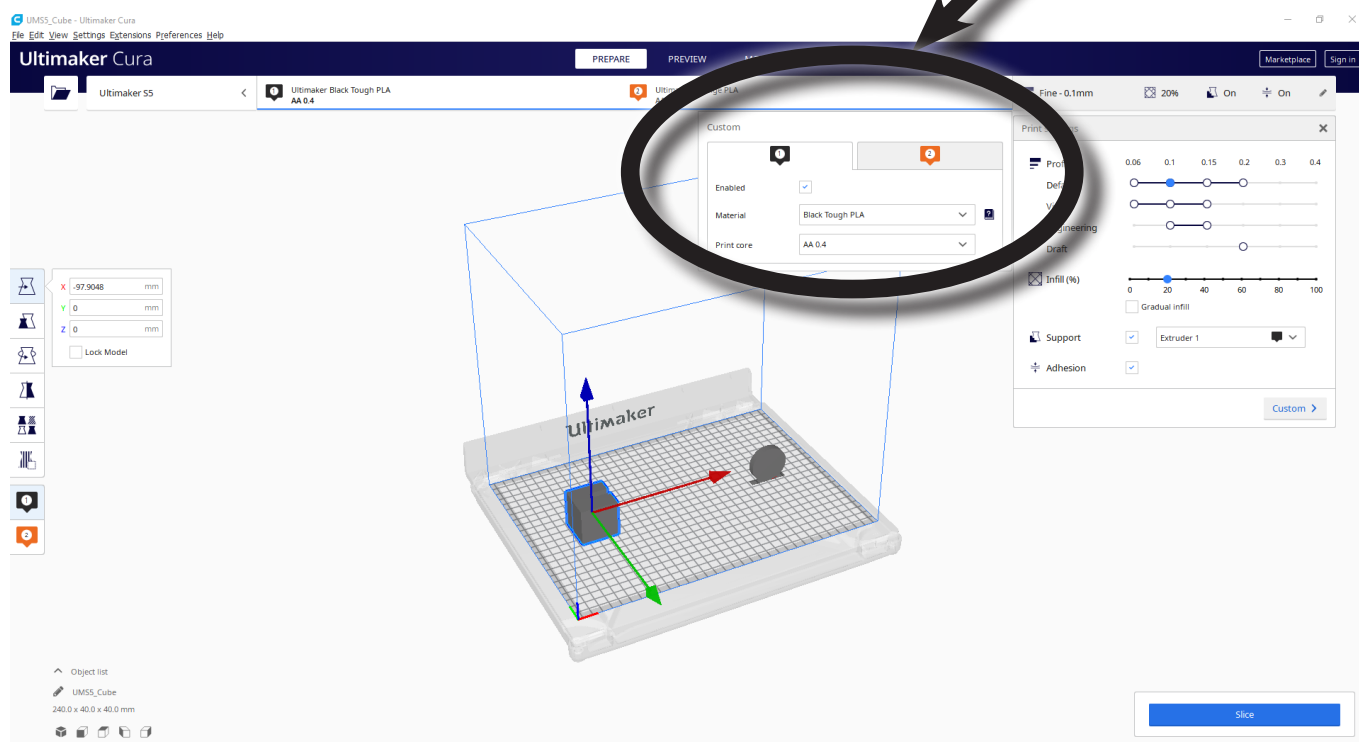
3. You should see your models in the virtual build platform. Click on the first model that you want to assign Hot End/Extruder 1. The Model will be highlighted when you do so.



4. On the left toolbar, select the Hot End you wish to use for this component of your print. Your Hot End selection will be shaded

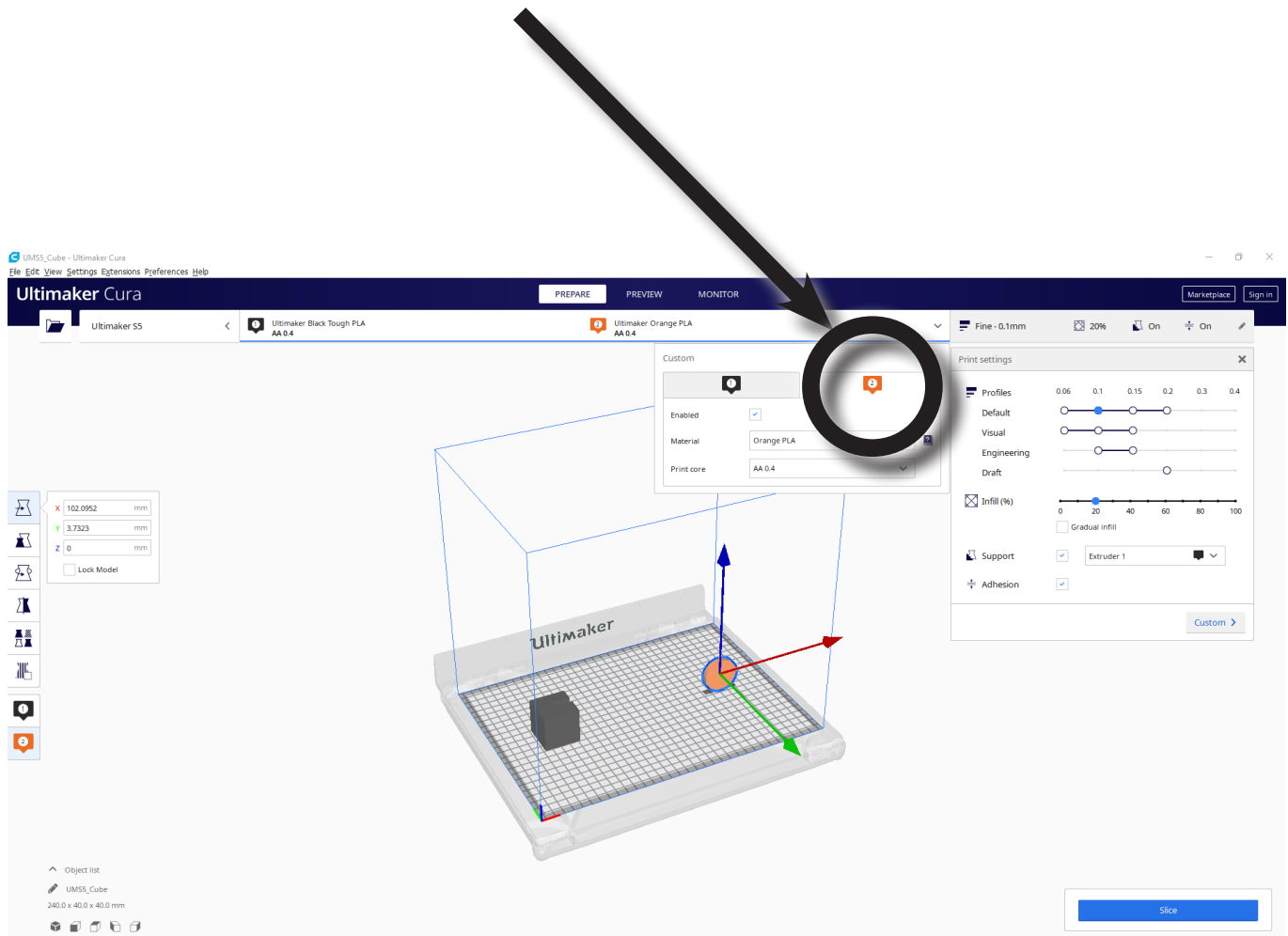


5. In the top toolbar, make sure that the materials and Print Cores are correct and that the “Enabled” box is checked

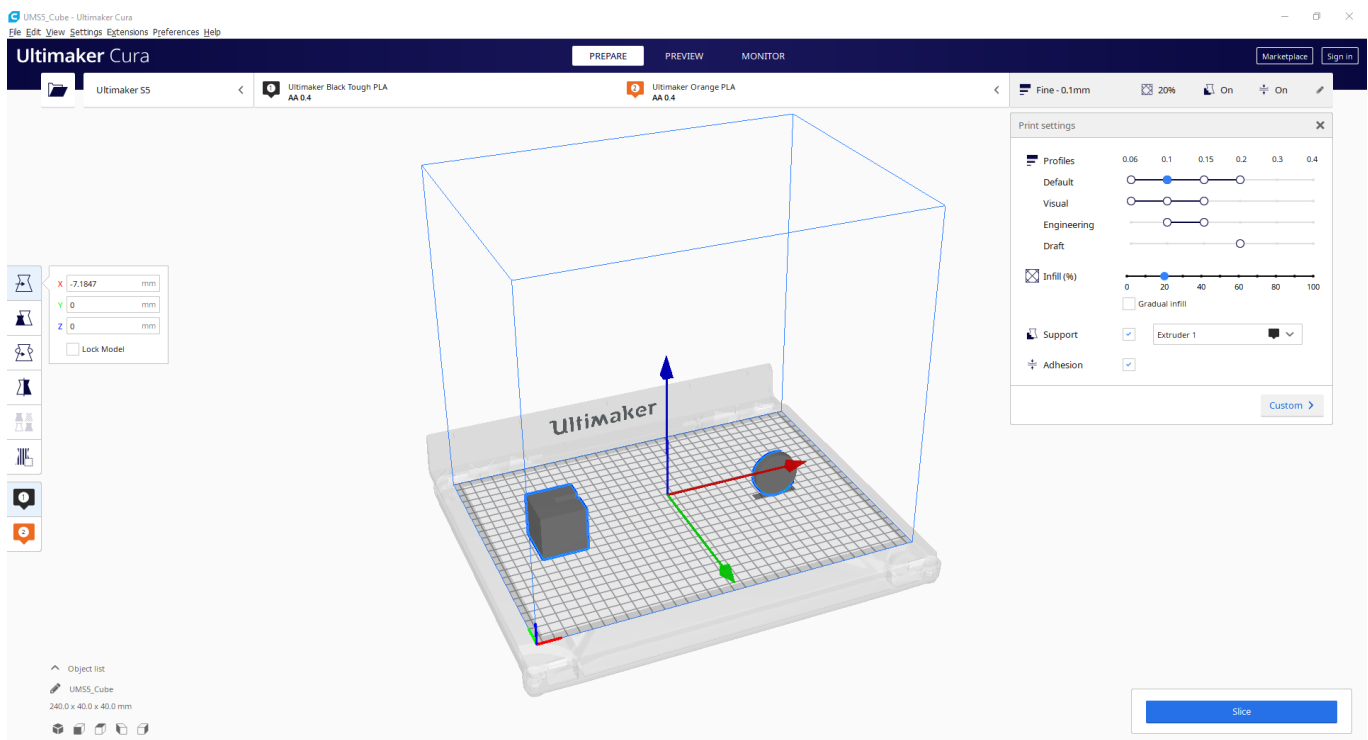




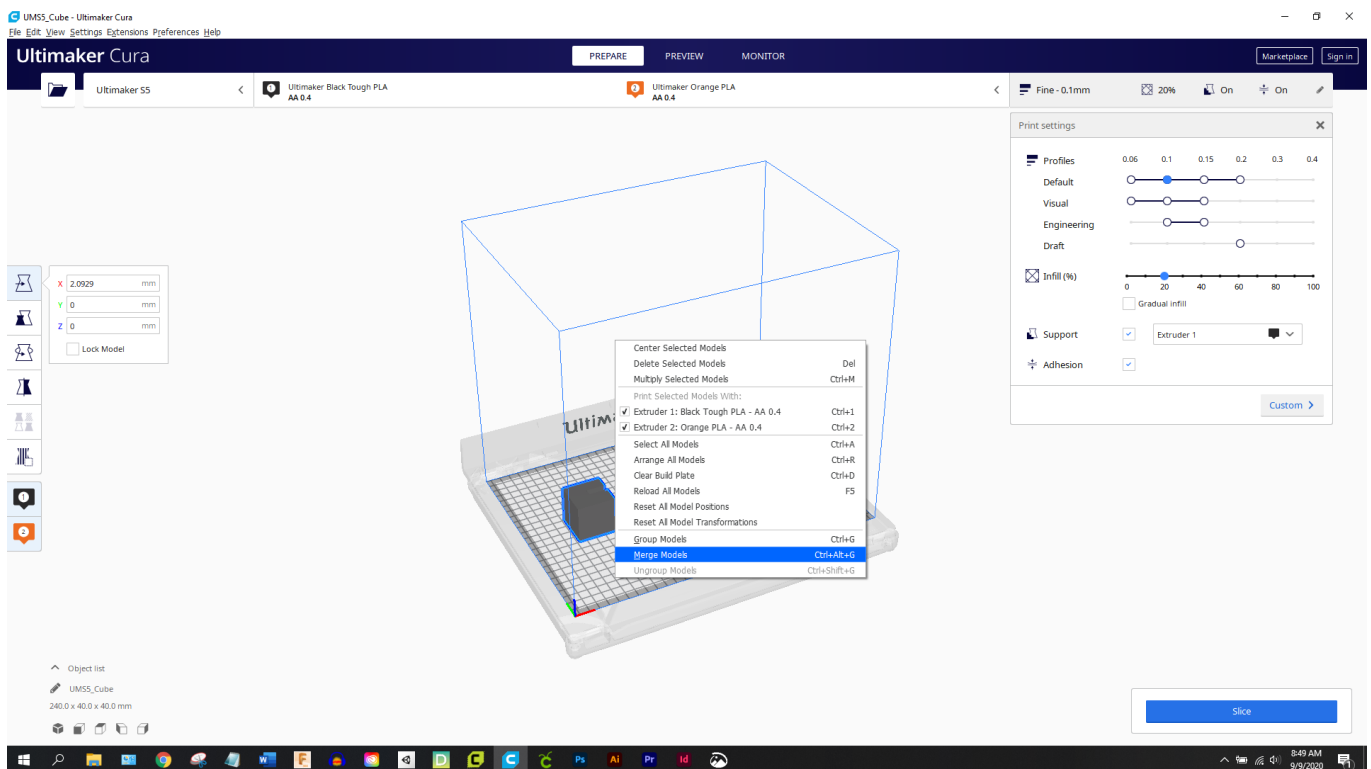
6. Repeat steps #2-4 using the second model, but make sure this time you select Hot End/Extruder 2.



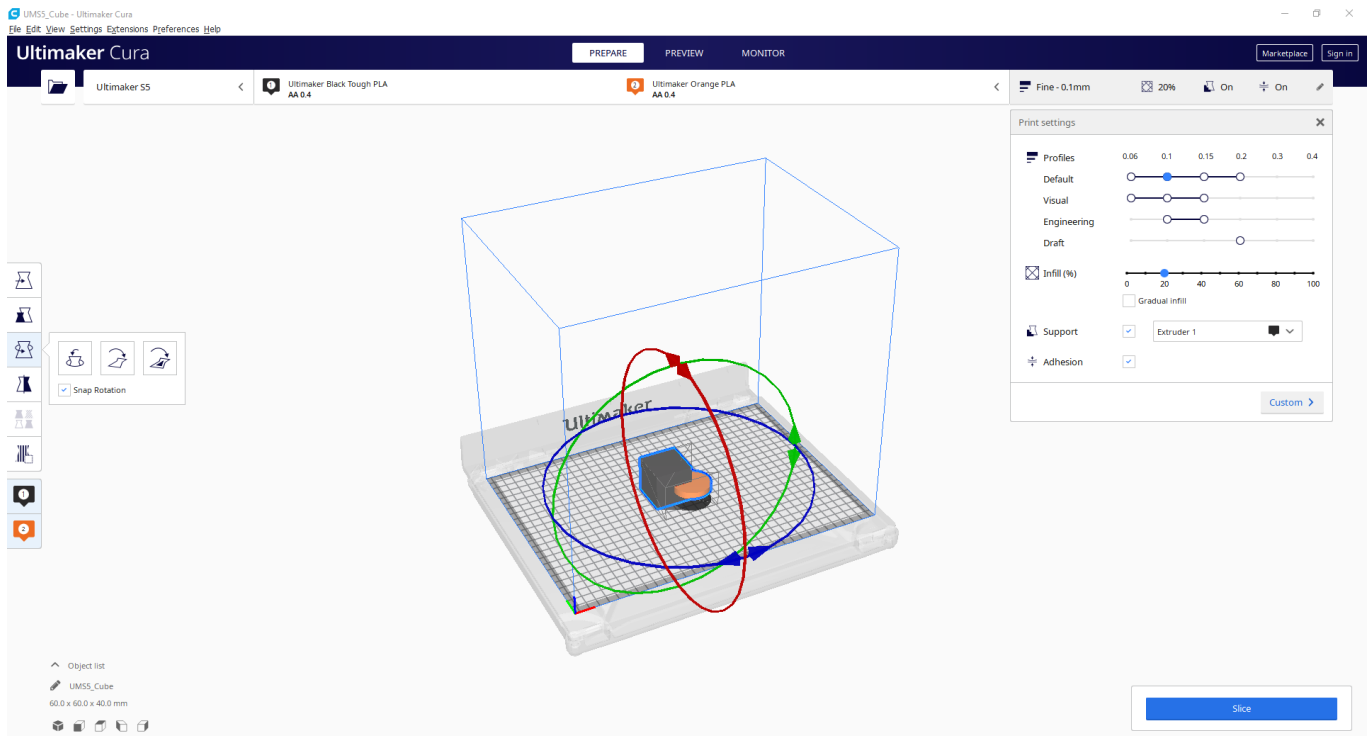
7. After you have assigned your Hot Ends/Extruders to the correct components, press “Ctrl & A” on your keyboard to select all. (Command & A on a Mac).



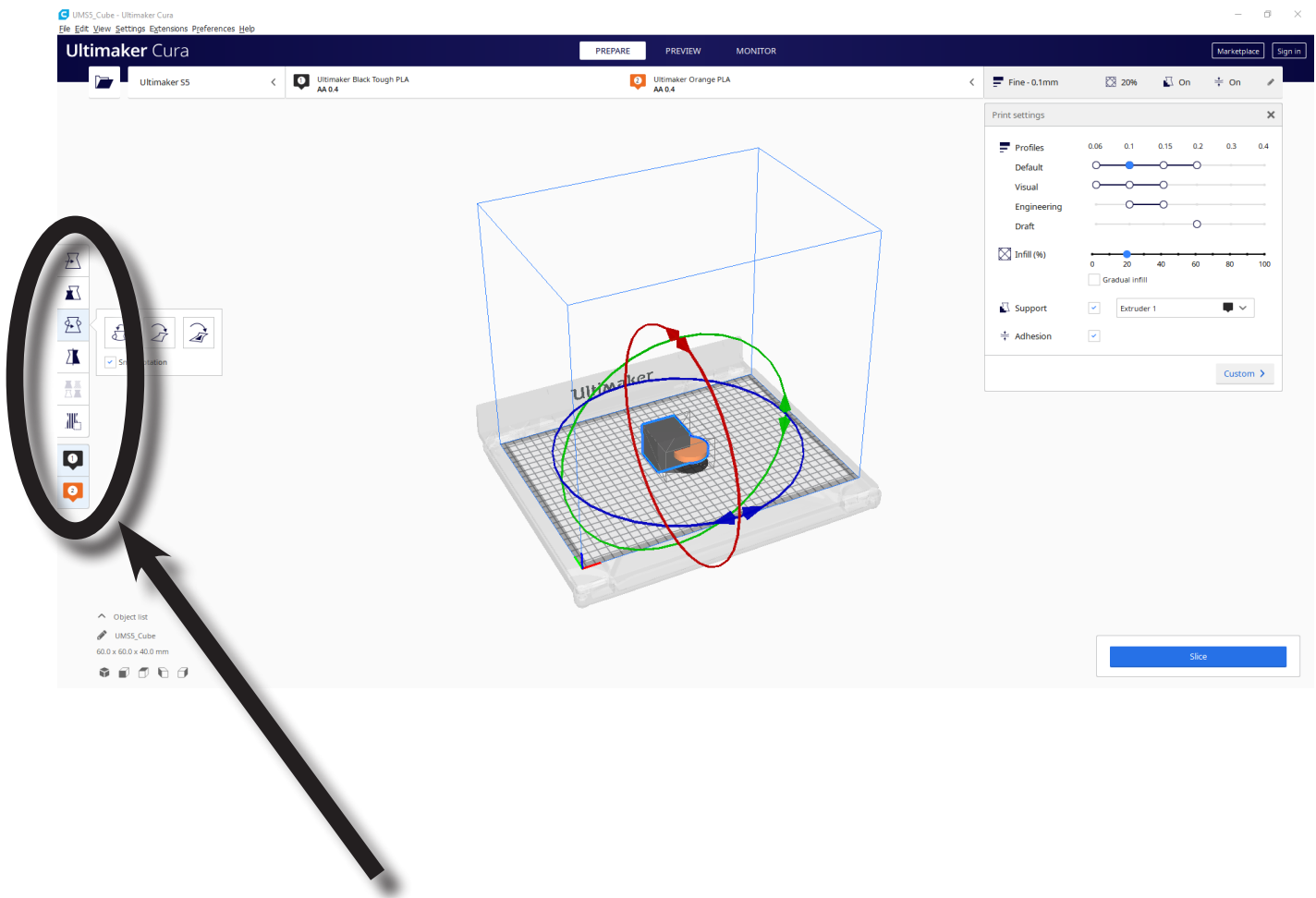
8. While all models are selected, right-click on the screen. This will bring up pop-up menu.



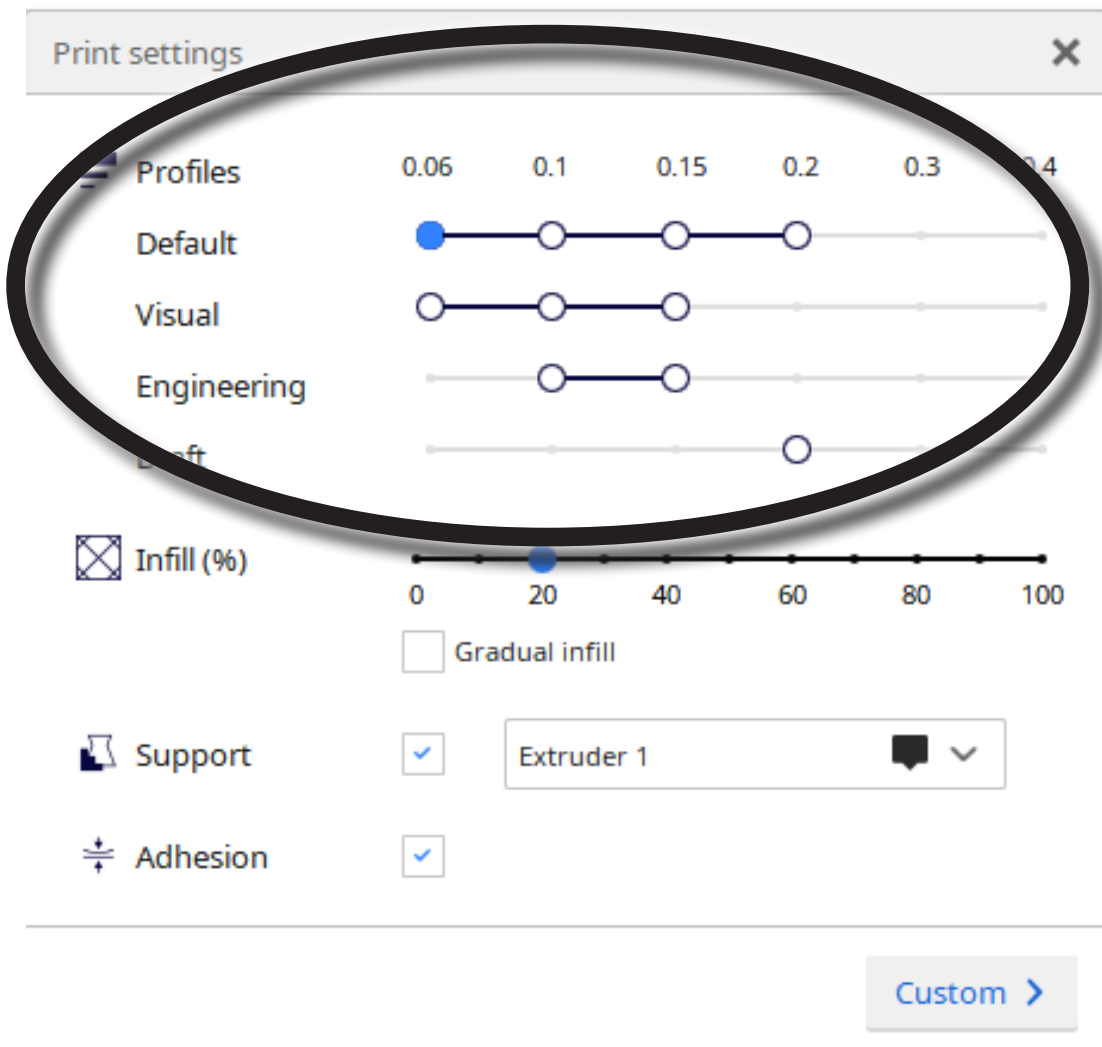
9. Click on the selection “Merge Models”. This will merge your models exactly how they were oriented when you exported them from your CAD program. NOTE: If they are not oriented correctly after you merge them, go back to your CAD file and trouble shoot.



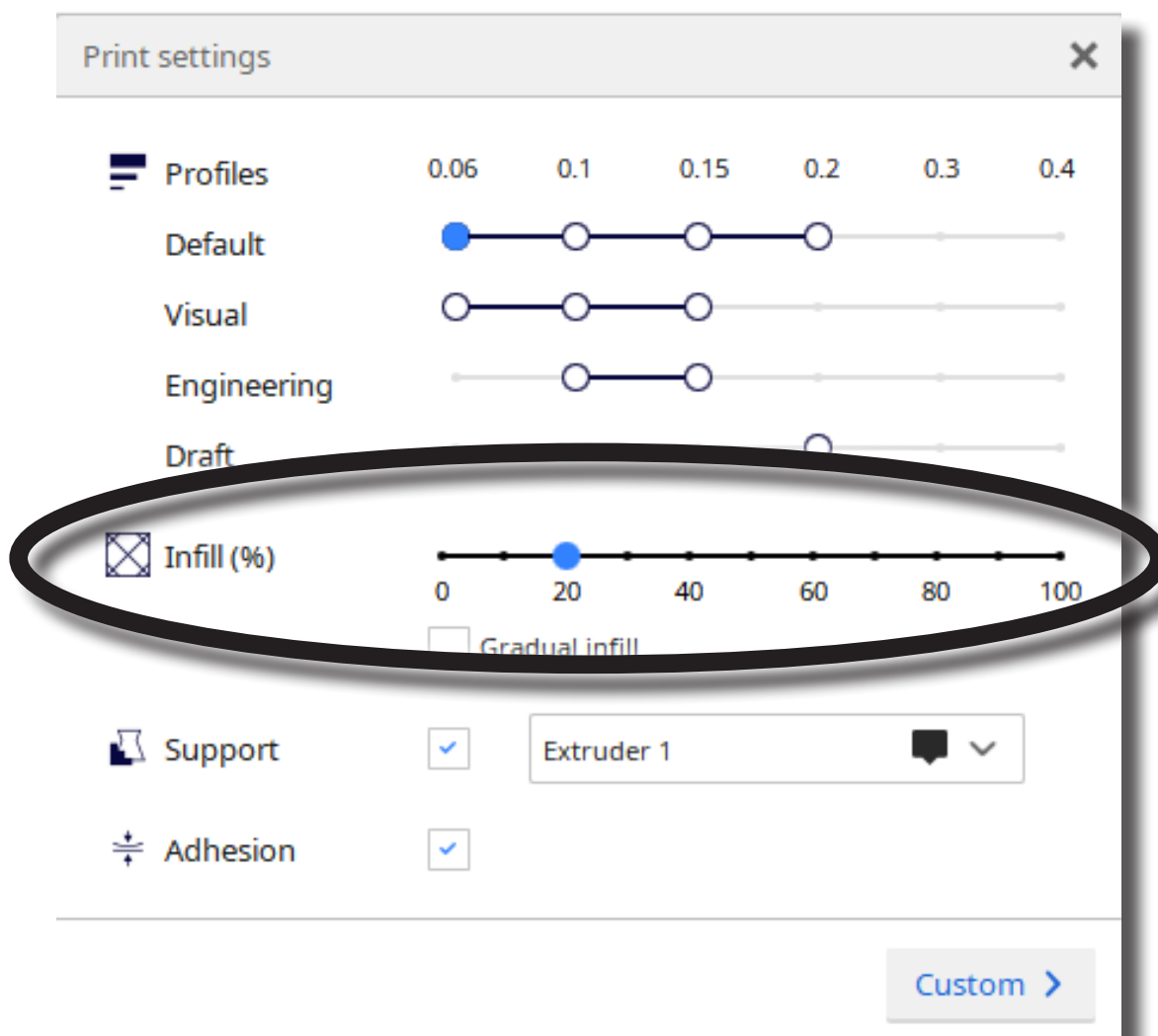
10. After your models have been properly merged, use the icons on the left toolbar to adjust your model (Scale, Rotate, Move, etc.)



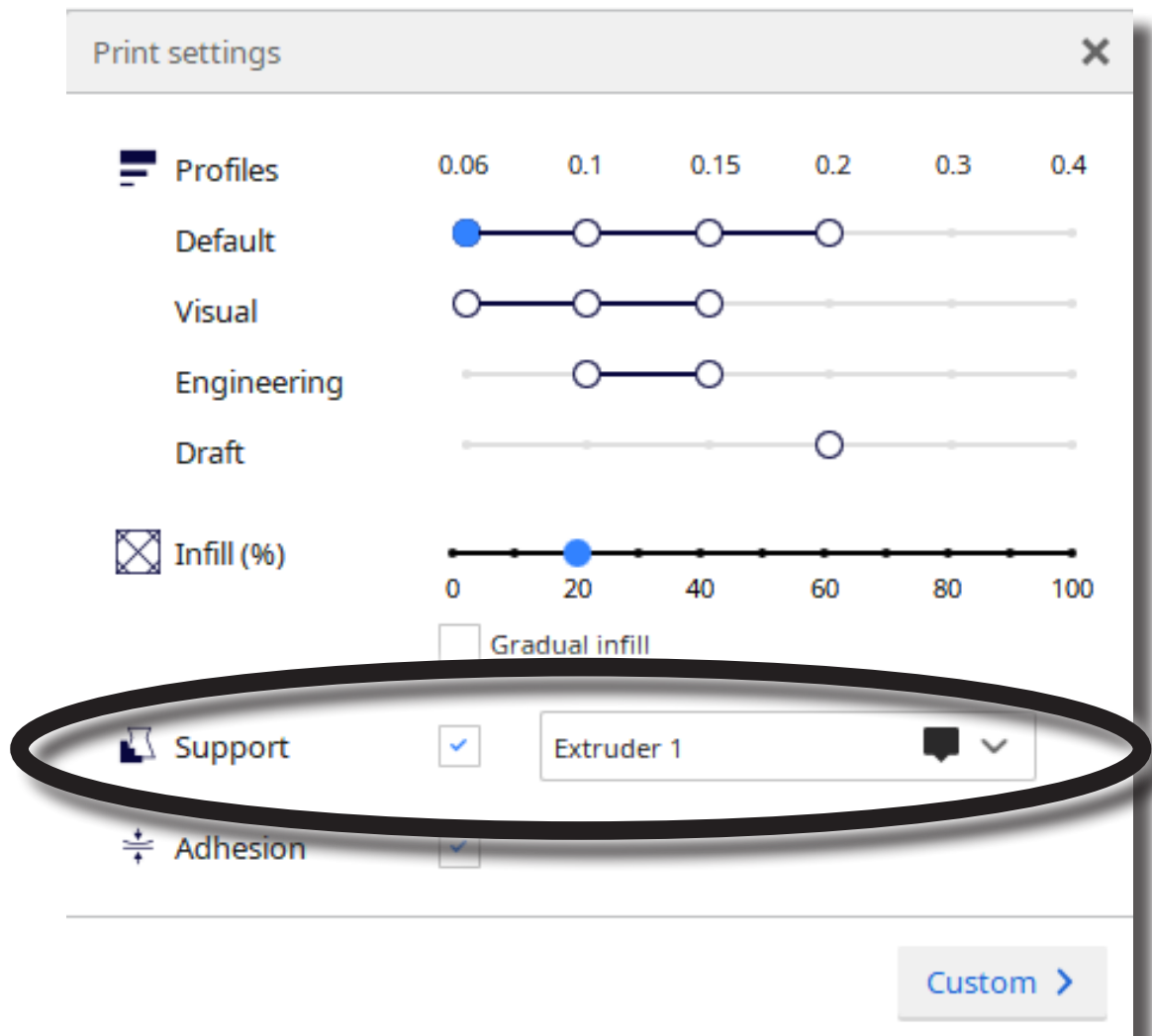
11. In the right toolbar, select your Profile. This is the print resolution. Keep in mind the trade-offs: higher resolution like 0.06 = longer print time, but better surface finish; lower resolution like 0.2 = Less resolution but a faster print time



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

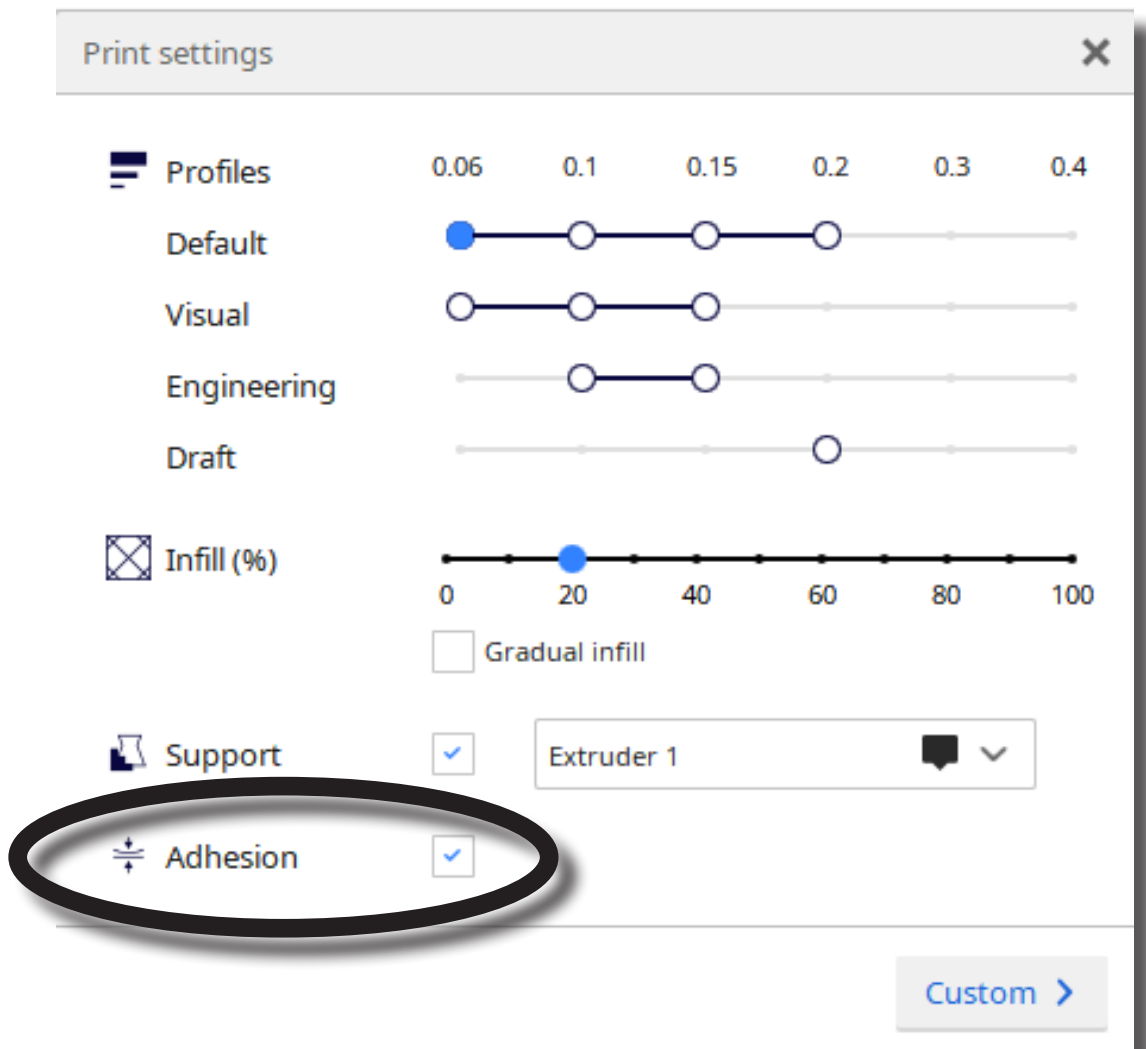


13. If your model has over-hangs, make sure to click the “Support” check-box. (NOTE: this will give you the option to use either Hot End/Extruder to print the support structure, so be mindful of this.) You can view the support structure by first slicing the model then clicking the “Preview” button at the top of the screen

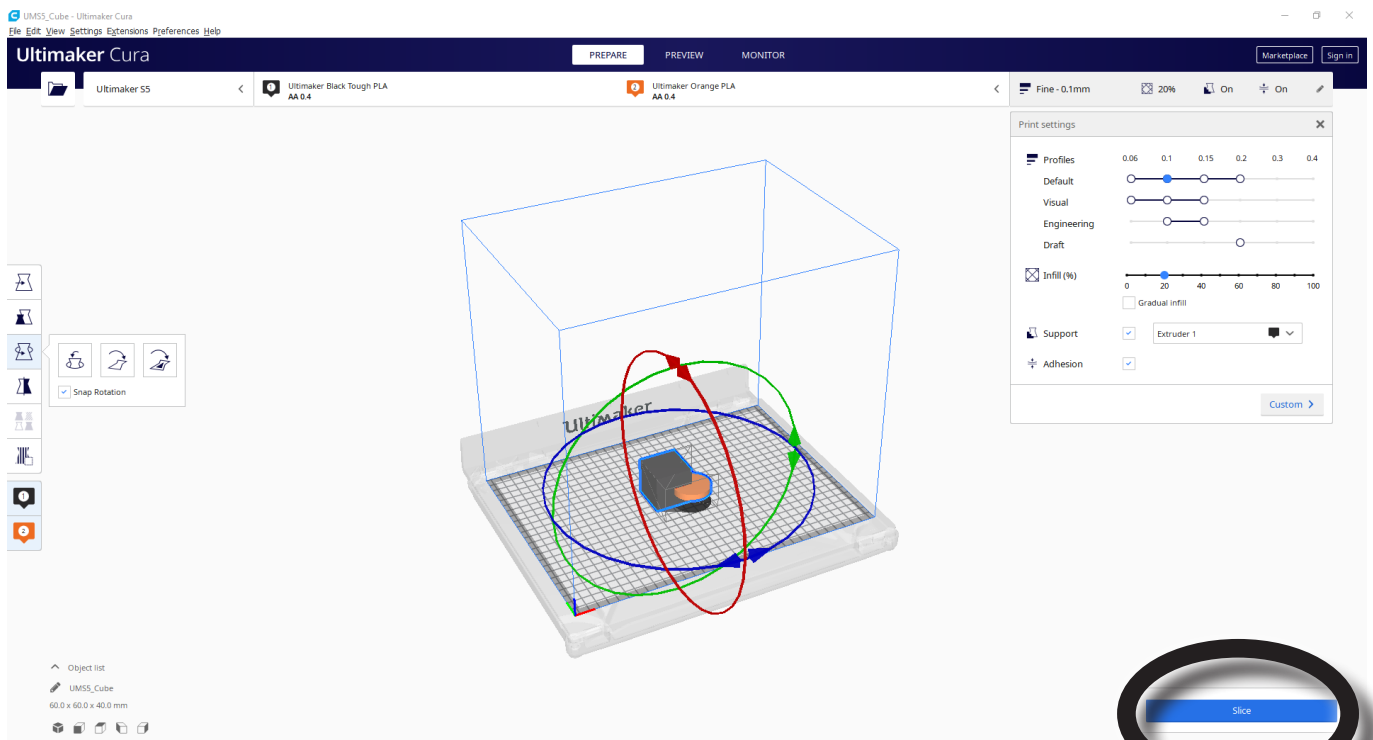




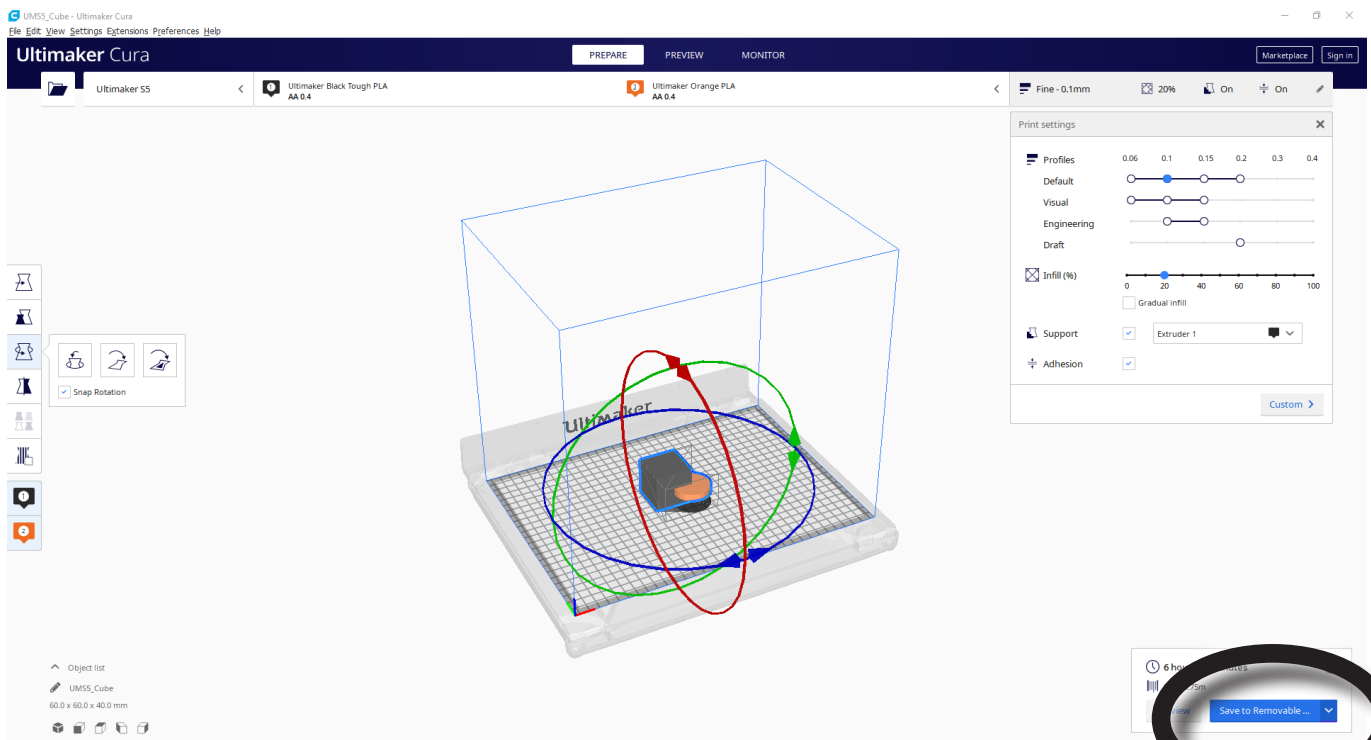
14. In the right toolbar, you can add an “Adhesion” to your print. The default adhesion type is a skirt, which places a single-layer-thick, flat area around your object, which resists the pulling forces as the print cools. (NOTE: It is NOT recommended to select “NONE”, as this increases the likelihood of a failed print due to the model “peeling” away from the build platform during the print).



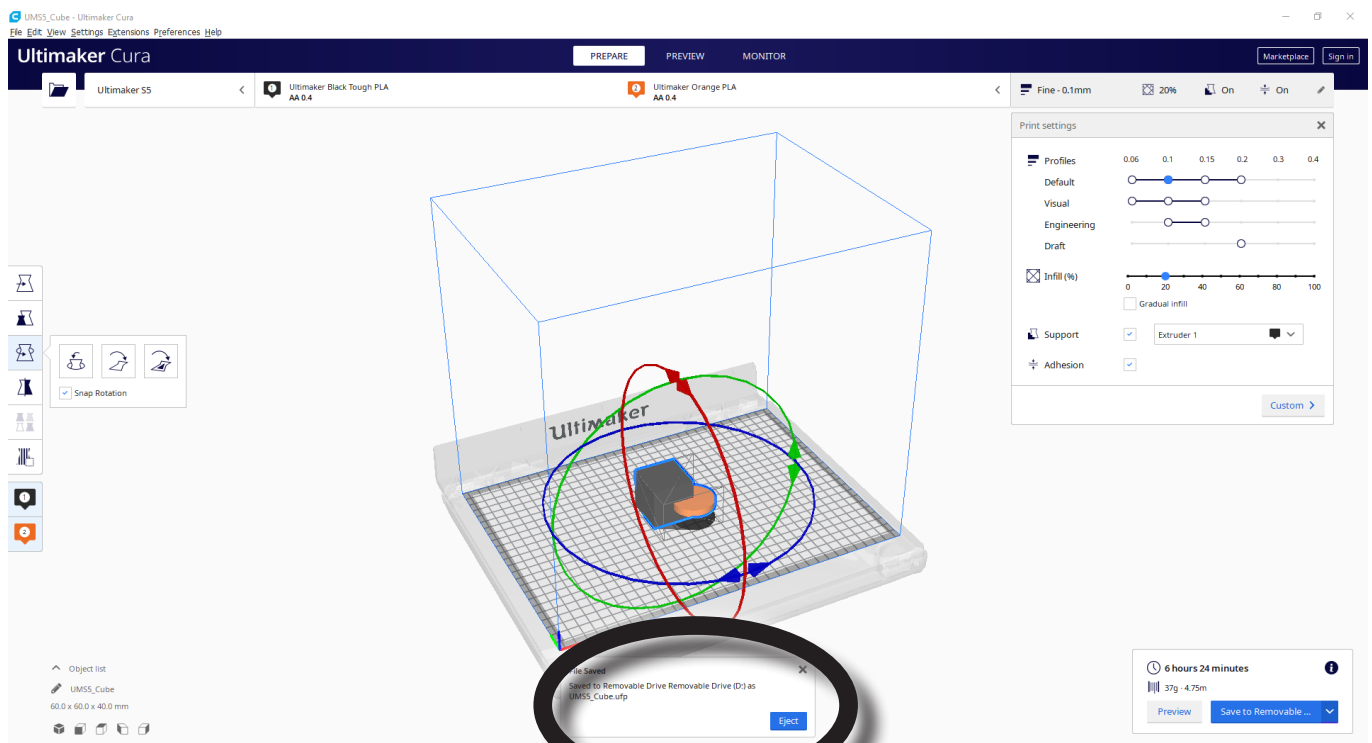
15. When you have finished adjusting the model and print settings, click the “Slice” button in the lower right part of the screen



16. After Cura finishes slicing the model, insert a USB drive into your computer then click the “Save to Removable Drive” button in the lower right part of the screen



17. In the lower center part of the screen, click the “Eject” button



18. Insert your USB into the USB port on the front of the printer



# MACHINE OPERATION

1. Ensure the correct filament is properly loaded into each extruder. If you need to check, you can tap the middle icon on the left part of the touch screen (looks like a printer). This will give you information on the what filament is loaded into each extruder and what type of hot end is being used for each extruder



2. Tap the "Print" icon (looks like a circle)





3. The touch screen should say "Ready to Print". Tap the "Select from USB" button



4. Locate the file you want to print by swiping up and down on the touchscreen and tapping the name of the file



5. Take a moment to look at the info on the screen. It will tell you the estimated time of the print, what extruder 1 will use for material, and what extruder 2 will use for material



6. If everything looks good, tap the “Start Print” button

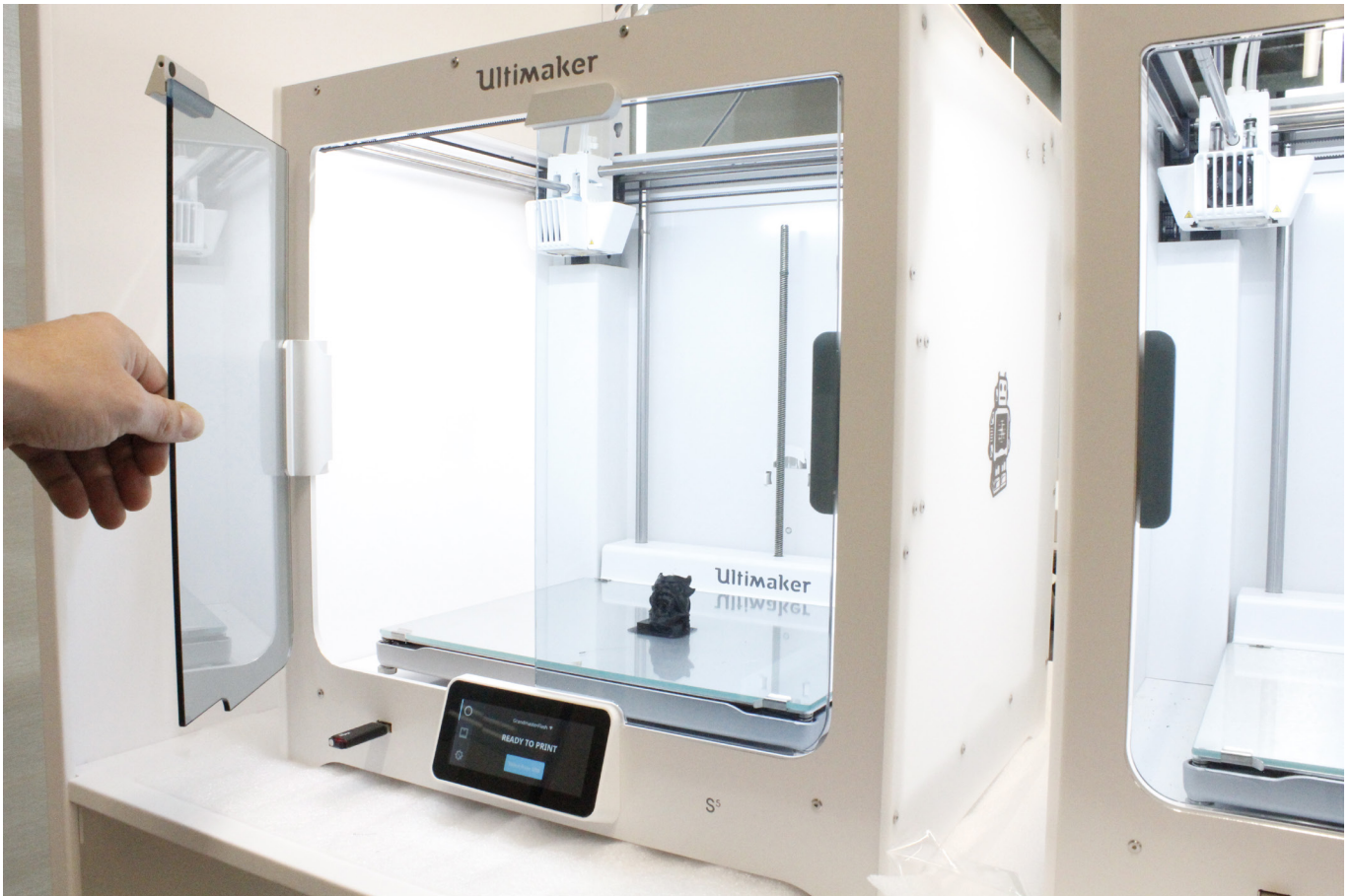


*NOTE: Both the extruder and the build platform may be very hot immediately after the print finishes. These components can burn you if you touch them. Therefore, wait a short period of time before removing your print. Doing so will help substantially when you remove your print from the build platform*

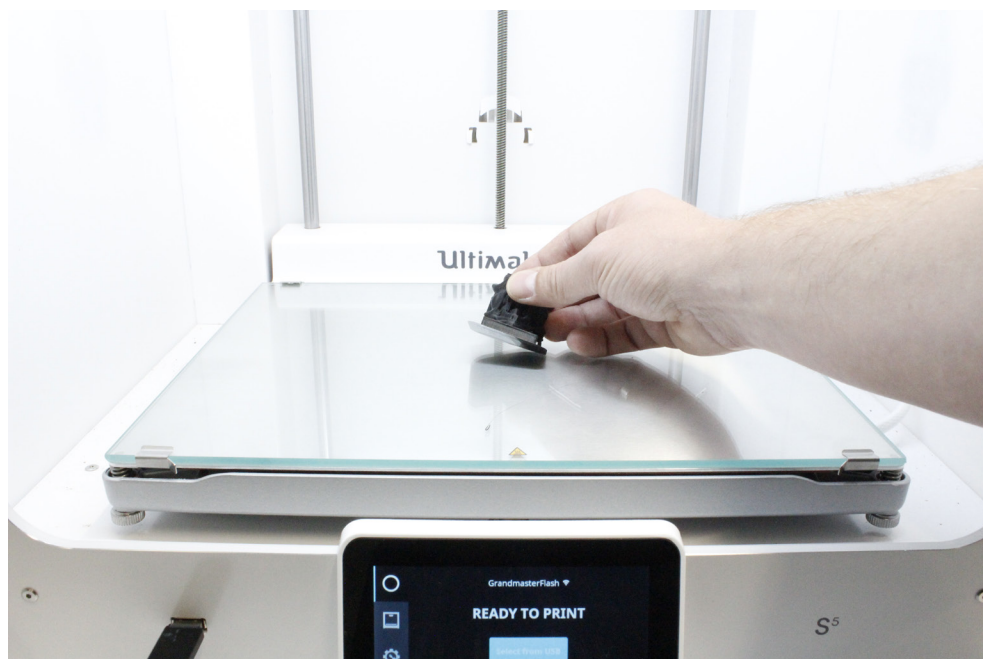
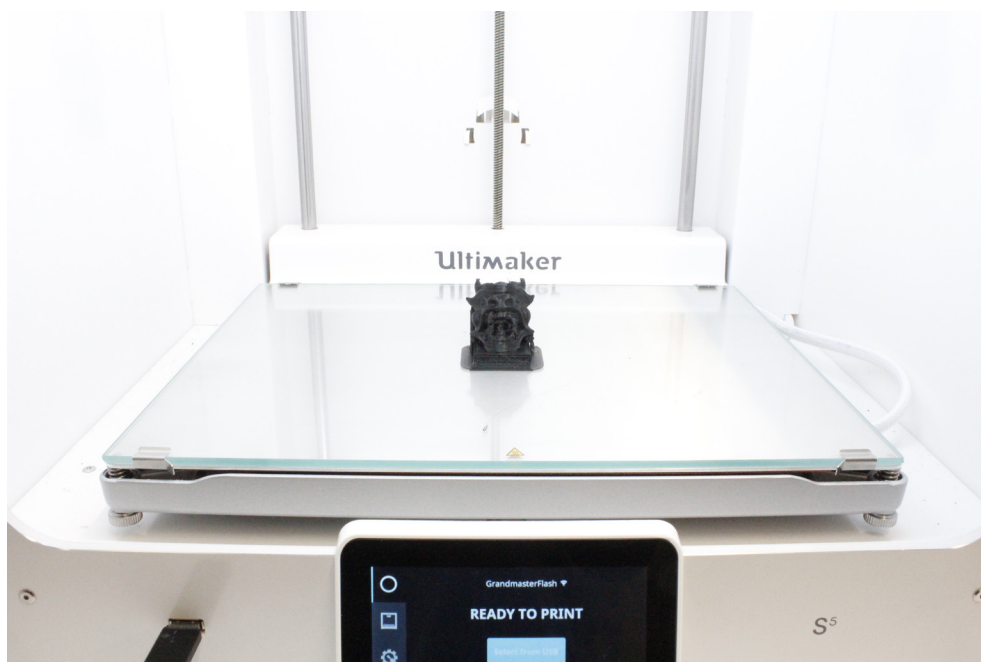
1. The touchscreen will have a green check-mark on it. Tap it to continue



## 2. Open the front doors

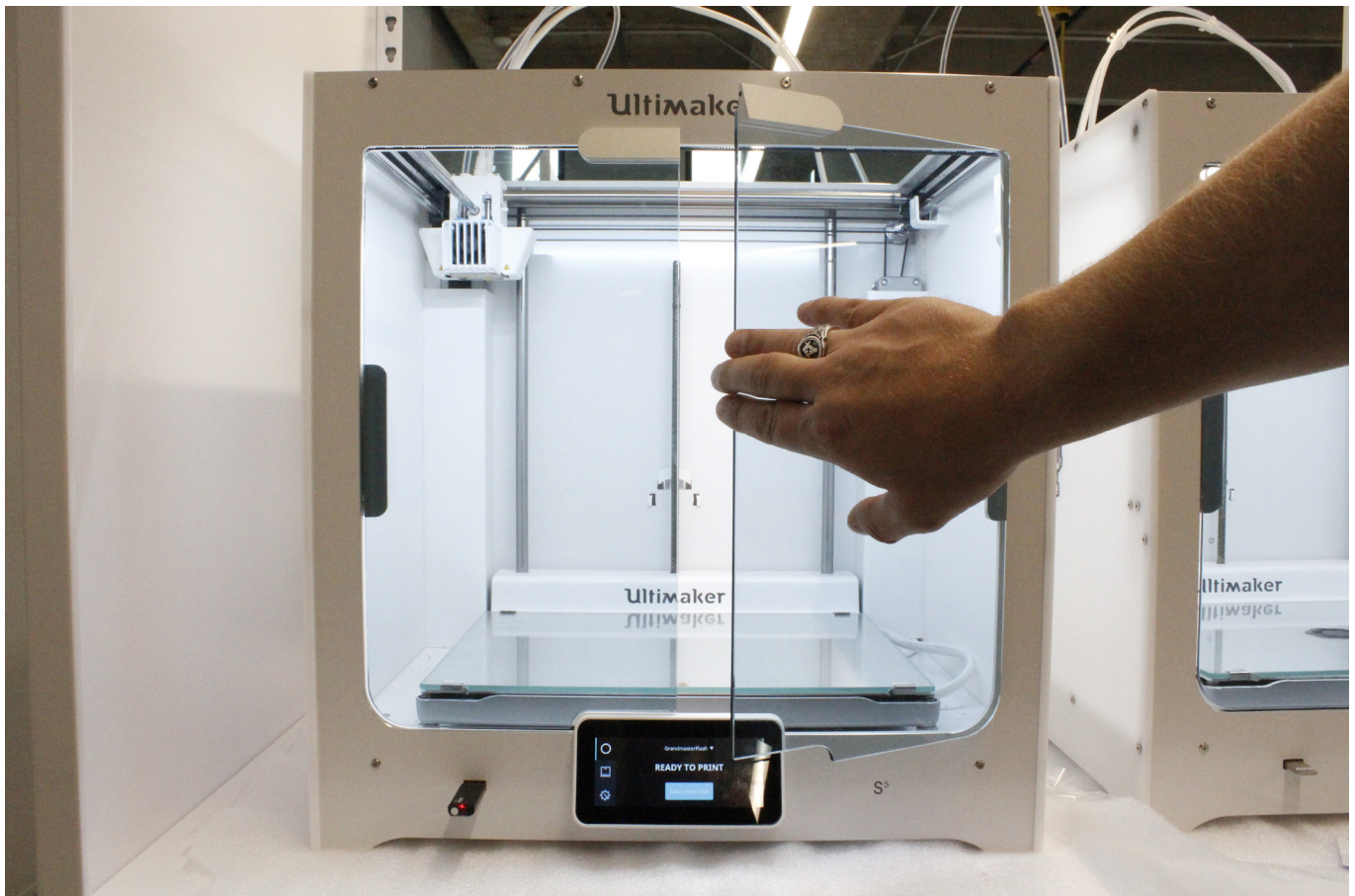


3. Firmly remove the print from the build platform. (NOTE: The print should come off easily. If the print does not come off easily, ask a Shop Technician for assistance, as you could accidentally damage the printer if you exert too much force on the print/build platform)





4. Remove any leftover filament from the build platform and close the doors



5. On the touchscreen, tap the “Confirm Removal” button

