STEP 1: CURA SOFTWARE

STEP BY STEP: PREPARING 3D MODELS FOR PRINTING

PUBLIC
FRISCO PUBLIC LIBRARY

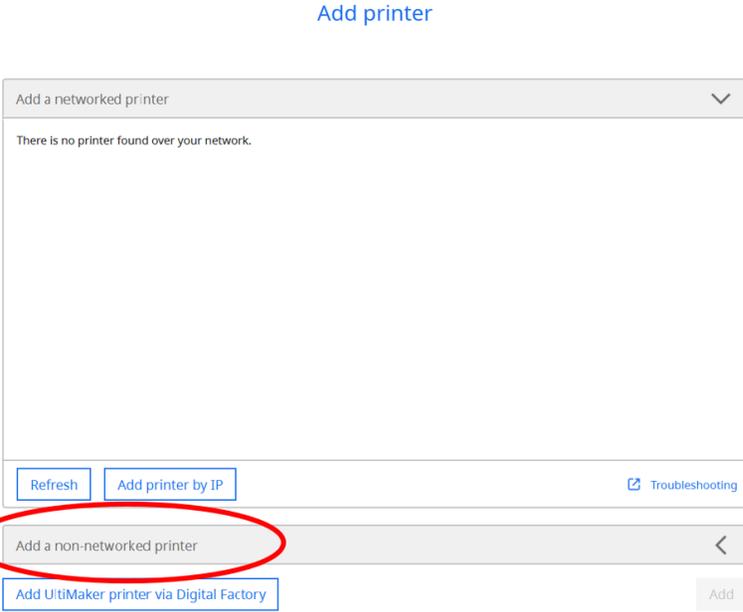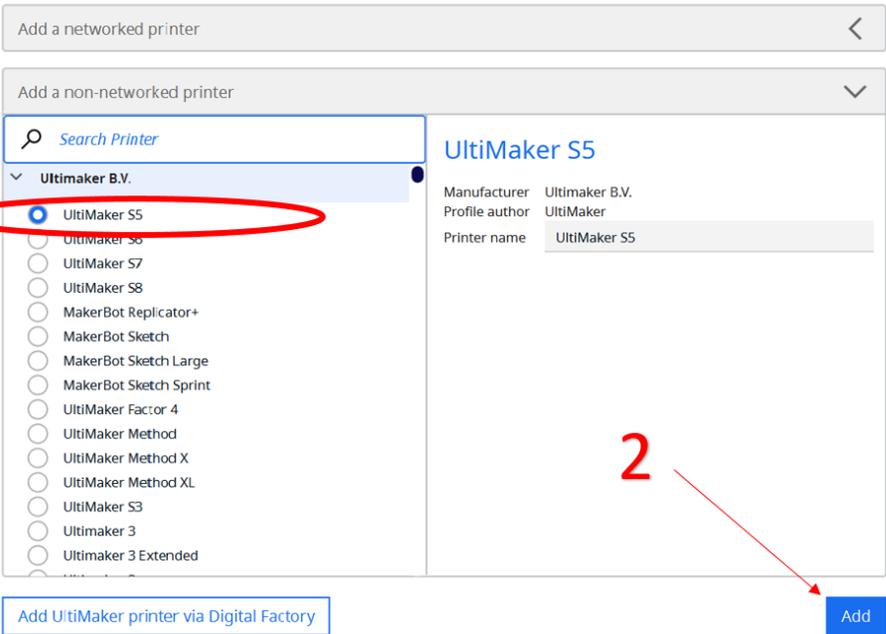| #	Steps	Key Points	Details	Pictures
1	Open Cura and Select Printer	Click the Add Printer option.	Ensures that the physical dimensions for the build plate are correct.	
2	You will be adding a printer.	Click **Add Printer**		
3	Select the type of printer	Select **Ultimaker Printer**		
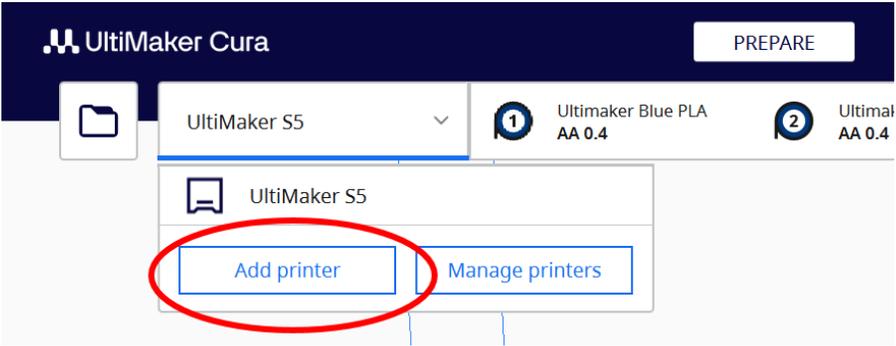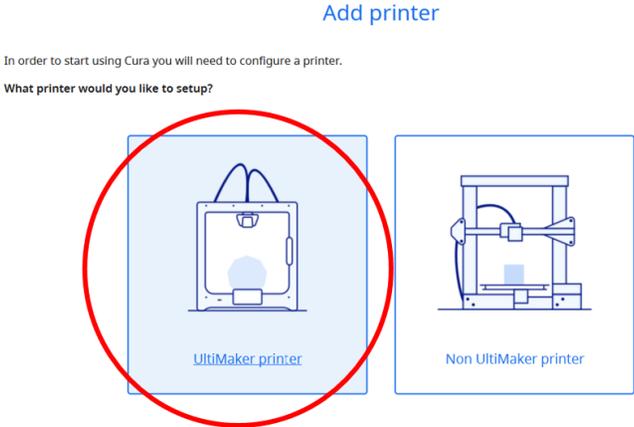
4	Setup for local printer use.	Select **Local Printer**	You will not need a DigitalFactory account. This feature is not supported at the library.	
5	Choose a non-network option.	Select **Add a non-networked printer**.	The printers at the library will not be accessible via a network connection. You will want to set up a non-network option.	

6	Select the model: **UltiMaker S5**	Check the radio button next to the model: **UltiMaker S5** Then click the add button in the lower right.		
7	Double check the correct printer is selected.	The printer model being display should be: **UltiMaker S5**	If you have a different model then the print preparation you do will not work when you attempt to print.	

8	Select Filament Color	Select **Extruder 1**	Ensure the correct color for print job.	
		2. Use Material dropdown to select **Ultimaker**.		
	Color availability:		All of our 3D printers default to using PLA in extruder 1 and do not use extruder 2.	
	• Red	3. Select **PLA**		
	• White			
	• Silver Metallic	4. Select color		
	• Black		You will not be able to change this configuration.	
	• Blue			
			We do not support 2 color printing.	
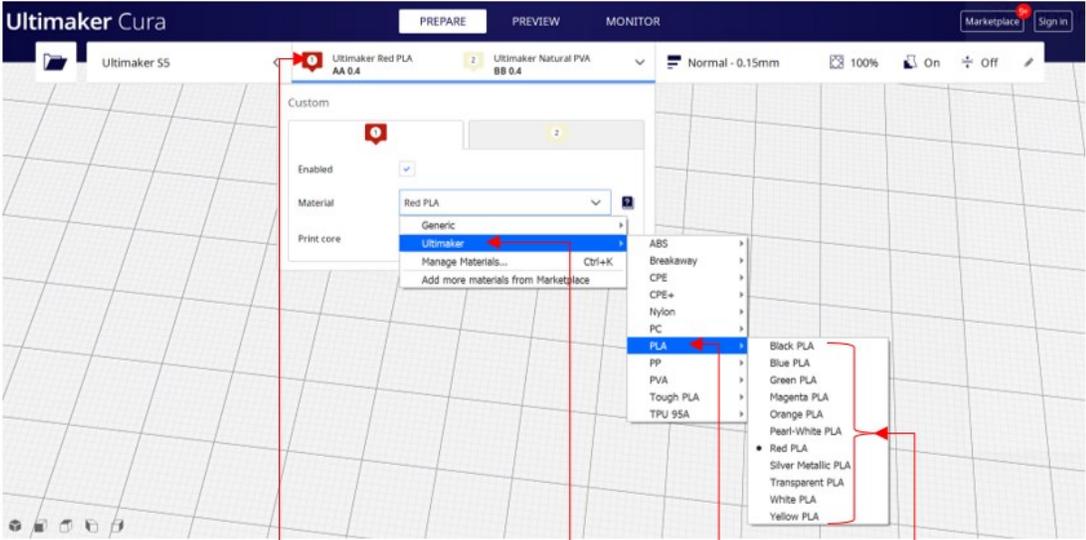
2.1   Select Extruder **1**

2.2   Use the material dropdown to select **Ultimaker**

2.3   Select **PLA**

2.4   Select color

9	Disable Extruder 2	1. Click on the extruder dropdown menu.	This will prevent error messages at the time of printing. The printer will know no material should be loaded into the #2 extruder.	
		2. Select extruder 2 tab.		
		3. **Clear** the Enabled checkbox.		

10	Check Print Core Designation	Make sure the print core selected is: **AA 0.4**	Different print cores have different performances. This lets Cura know exactly the type of print core you will be using at the library.	
11	Open 3D files	File needs to be a 3D model file. 3D Model File Formats: • .OBJ • .STL MANIFOLD ERROR MESSAGE? This object cannot be successfully printed.	Brings model into Cura software to be prepared with print instructions for the printer. Checks that model is manifold (also known as mesh errors or water-tight check).	 3.1 Click the file folder icon 3.2 Navigate to your .stl or .obj file 3.3 Select your file and click **Open**
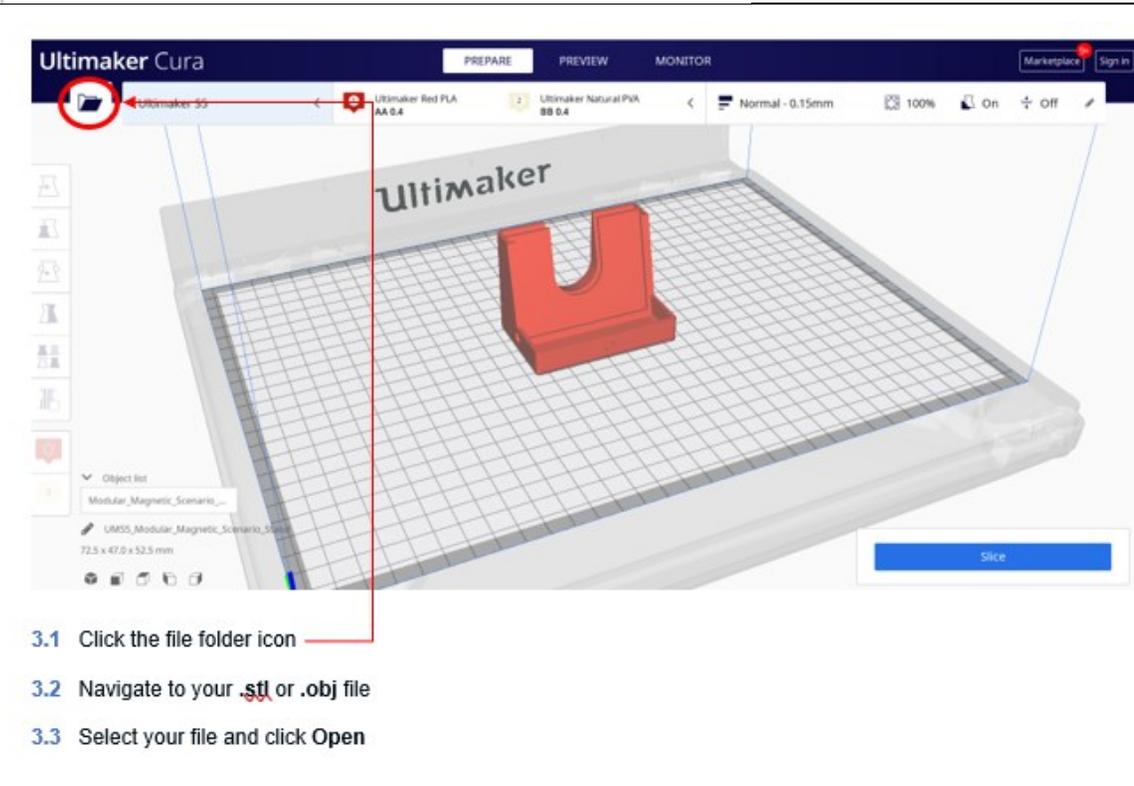
| 12. | Optimize Orientation of Object | Determine best orientation to minimize any overhangs.

Best practice is to put a large surface on the build plate of the printer (gray grid). | Print job could have major printing errors and be significantly more expensive if not done properly. | |
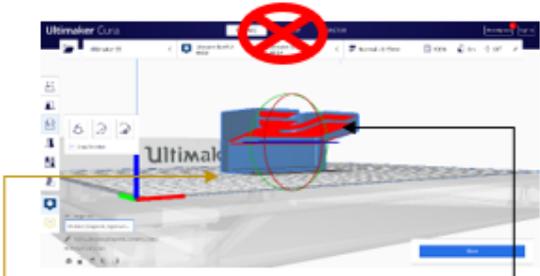
Unnecessary overhangs (indicated in red)

Small surface area touching build plate (gray grid)
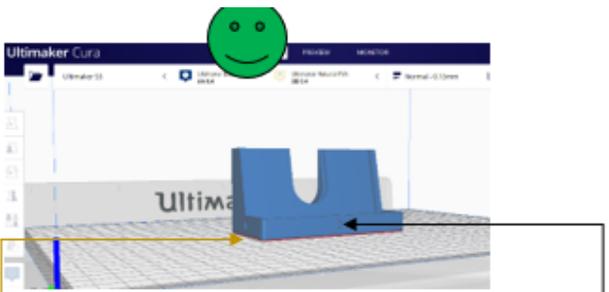
Minimized overhangs (only small areas highlighted in red)

Large flat surface touching build plate

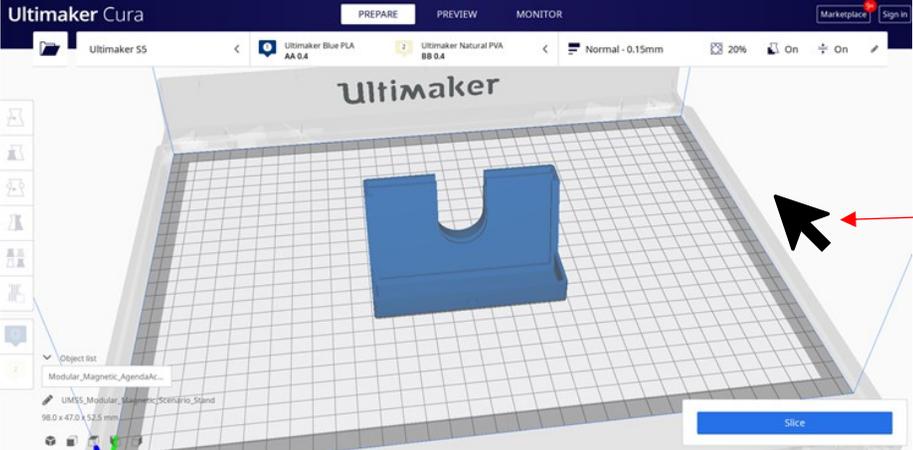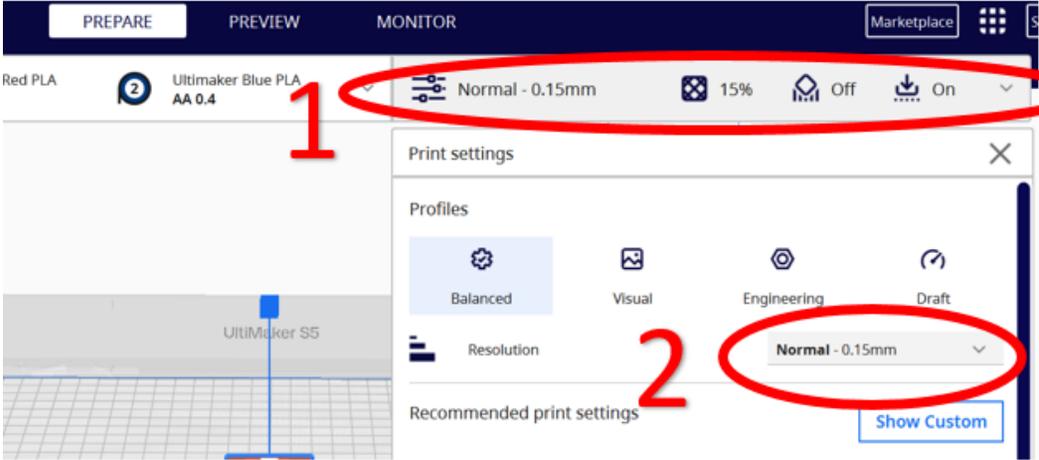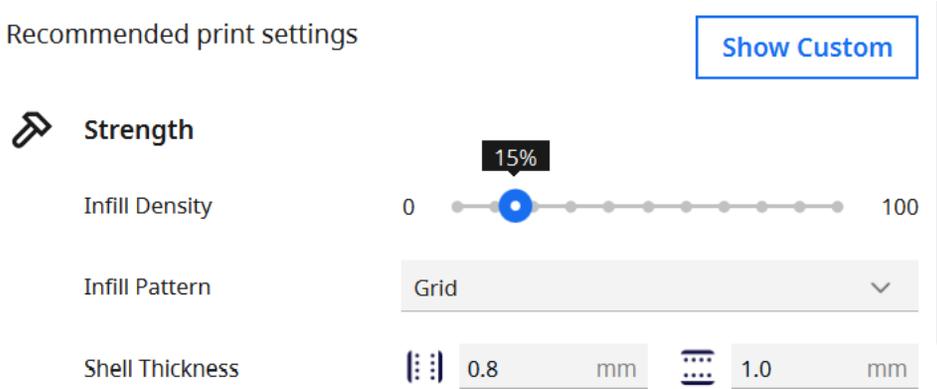| 13.	How to Change Orientation	Click on the object to select the object		
	See Step 14 to resize object.	Click on "Rotate" button on left hand menu.		
		Left click and hold mouse your mouse click down on the desired orbit to rotate object. As you move your mouse the object will rotate on the selected orbit.		In this example we left clicked on the red orbit and while holding the left click down, moved the mouse towards the top of the screen:
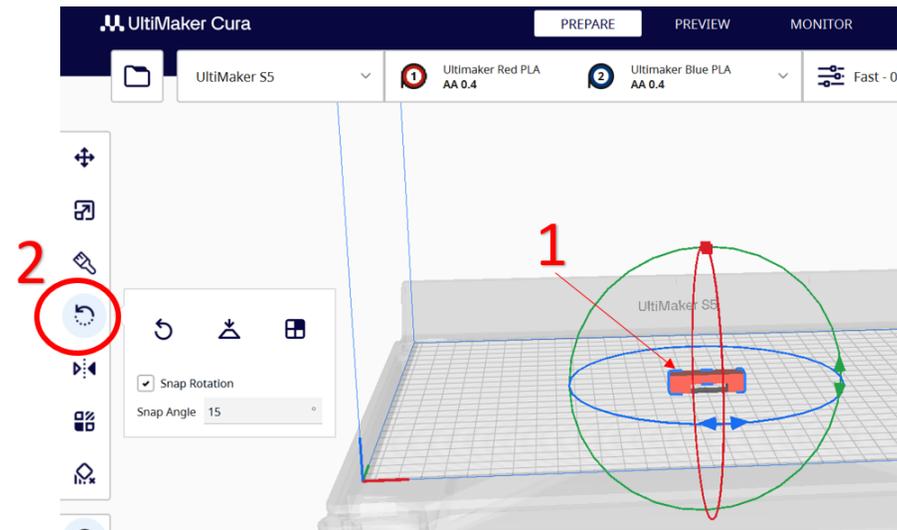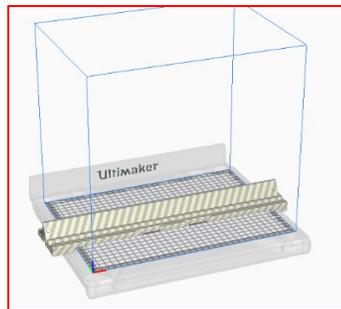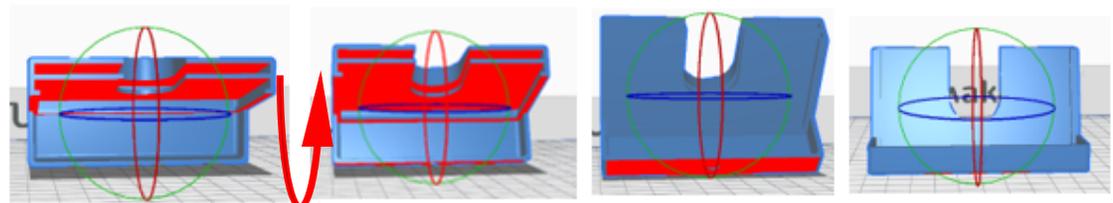
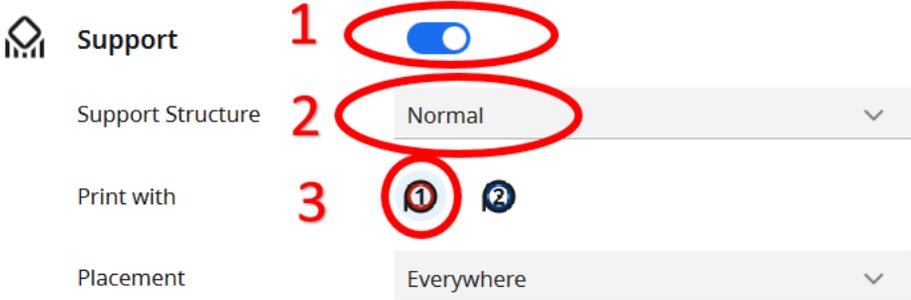A striped appearance indicates the object is too large to print.

Options include to:
- Resize the object so that it fits in the printable area.
- Re-orient the object so that it fits in the printable area.
- Redesign the object into smaller components that can be assembled after printing.
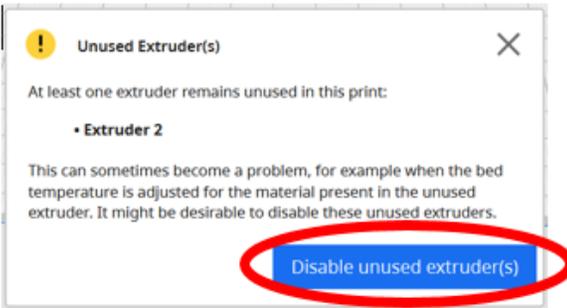
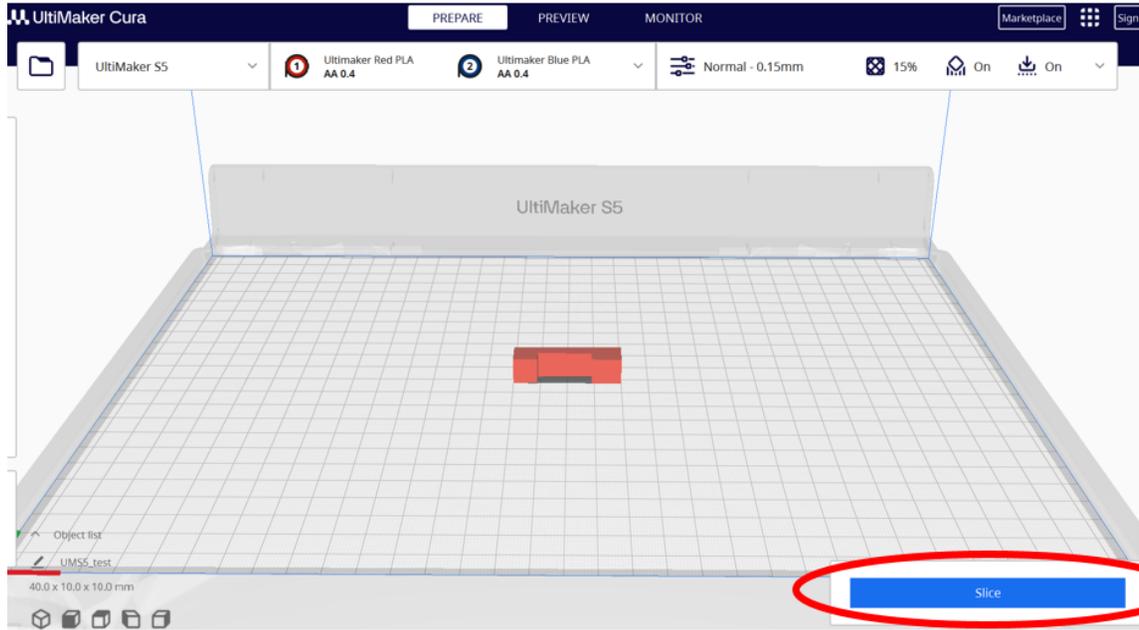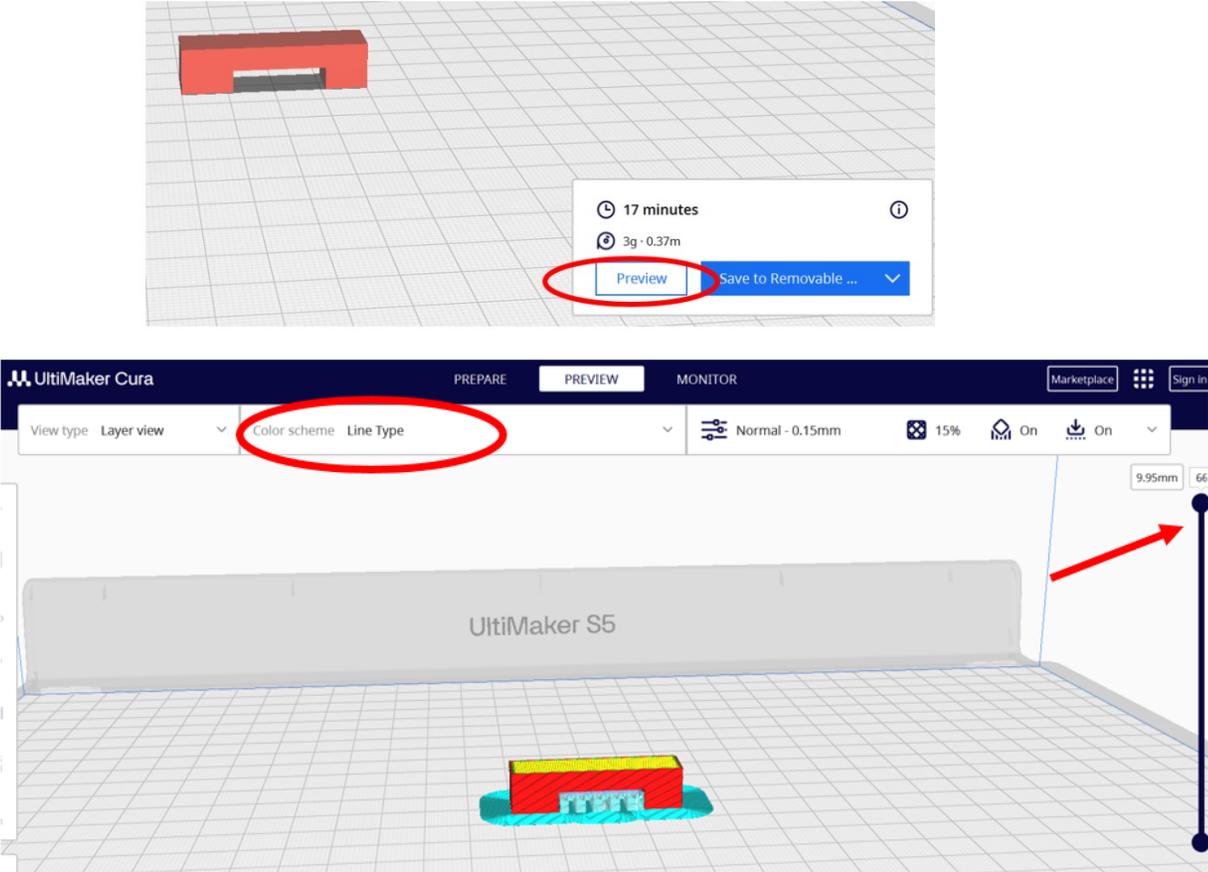| 14.	Review Object Orientation	Right click and hold the right click down in an empty space away from the object. While holding the right click down, move the mouse slowly from left to right to spin the view of the object to see the other sides.	Check for overhangs. Check that the object is flat against the plane.		Right click and hold the click while moving mouse.

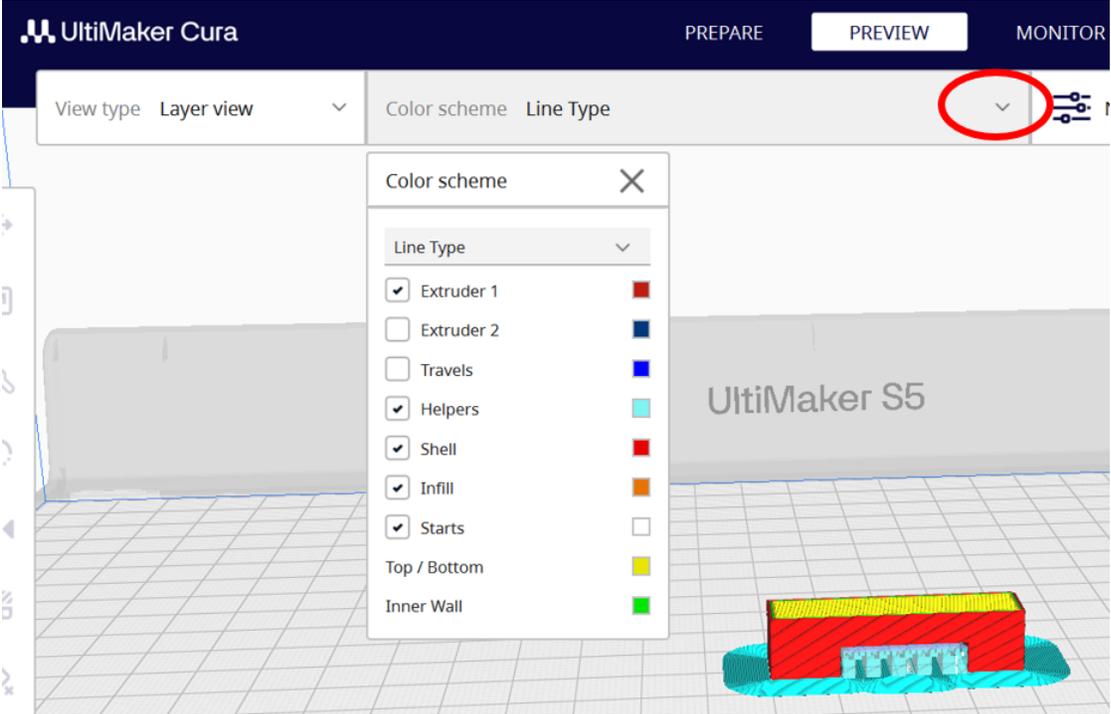| 15. | Re-size Object (If needed) | 1. Click on object.

2. Click on "Scale" button in left hand menu.

3. Check "Uniform Scaling".

4. Enter a new value for the X, Y, or Z dimension.

With uniform scaling the software will update the other dimensions to keep the object true to original proportions. | This is to scale the print job and make sure the print job is the correct size.

It is recommended to making sizing changes in the design software when possible and not in Cura.

Using a scaled increase or reduction can result walls and other design elements becoming thinner/thicker than desired as well as other unintended modifications that come with altering the overall size. | |

16	Update Print Settings: Profile	1. Click on Print Settings bar. 2. Select either: Normal **.15mm** resolution Fast **0.2** profile Reduces print time but also makes each layer more visible	Alters the layer height of each print level. Other resolution options are not supported at the Frisco Public Library.	
17	Update Print Settings: Infill Density	0 – 10% **Not recommended** most likely to give poor end results. **15 – 30%** recommended for most items **40 – 60 %** very strong, slow print time, good for parts under stress **100% solid** slowest print time, highest cost	This will change how dense your solid areas will print. Increasing the infill density increase the amount of material used, weight, strength, and print time.	

18	Update Print Settings: **Turn On** Supports	1. Turn on supports 2. Select Normal 3. Choose **Extruder 1** for PLA supports.	Supports used to help provide scaffolding needed for overhangs. NOT CHECKING SUPPORTS IS THE MOST COMMON CAUSE FOR FAILED PRINTS.	
19	Update Print Settings: **Turn On** Adhesion	Turn on the Adhesion slider.	Adhesion will add a removable brim to help keep the object in place during the printing process. If the object moves during printing, the print job will not successfully complete.	

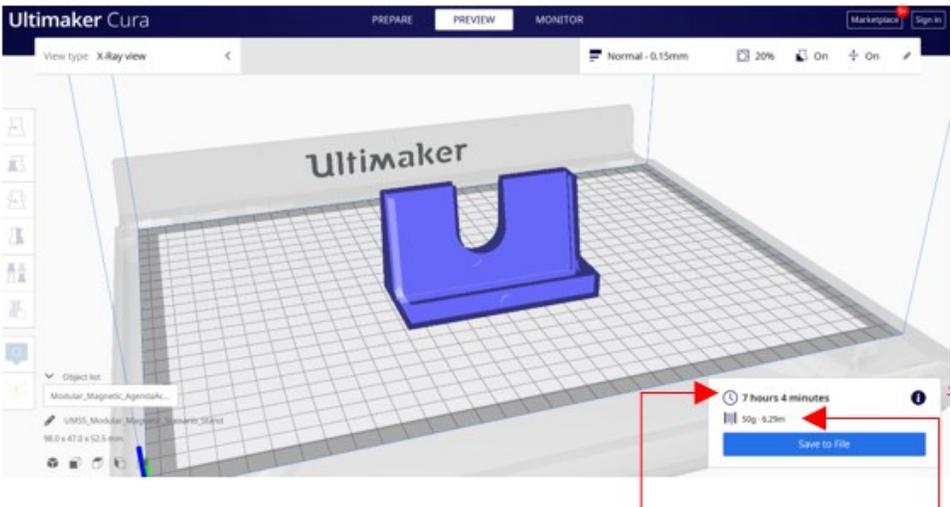| 20 | Slice | Click "Slice" button on bottom right.

If prompted, disable unused Extruder 2. | This will transform the file from a .stl or .obj file into a G-code file that includes layer by layer specific instructions for the printer type chosen.

The printer uses these instructions saved as a .**UFP** file to recreate the object. | |

| 21 | Preview | Click "Preview" button.

Change the **Color scheme** field to **Line Type.**

The line type preview is a color-coded preview that allows you to see exactly how the printer will create the 3D model.

Use the slider on the right to see the model layer by layer. | This creates a layer by layer view of what the printer will be able to print.

This allows you to check for issues (unprintable elements) and see how much temporary supports are required during the printing process.

A careful review of the Line Type preview allows you to identify problems before you print. | |

| 22 | Preview | Blue sections are the supports. | This will allow you to see how much clean-up work is required after the object is printed. | |
| | Review Supports | To see a preview that does not include the blue supports (this will allow you to view elements of the model hidden by the temporary supports), click on the dropdown next to line type. Then uncheck the Helpers. | **IMPORTANT**: Supports in small or difficult to reach places will be difficult to remove later. | |

| 23 | Preview

Review for unprintable areas | Make sure slider is up to the top position.

Any areas in gray are elements of the object the printer cannot recreate. | Some elements may be too narrow or thin for the printer to recreate.

The printer extruder nozzle is 0.4mm wide. Elements smaller than this cannot be printed. | |
|----|----|----|----|
| 24 | X-Ray Preview | Check for problem areas that are flagged in red.
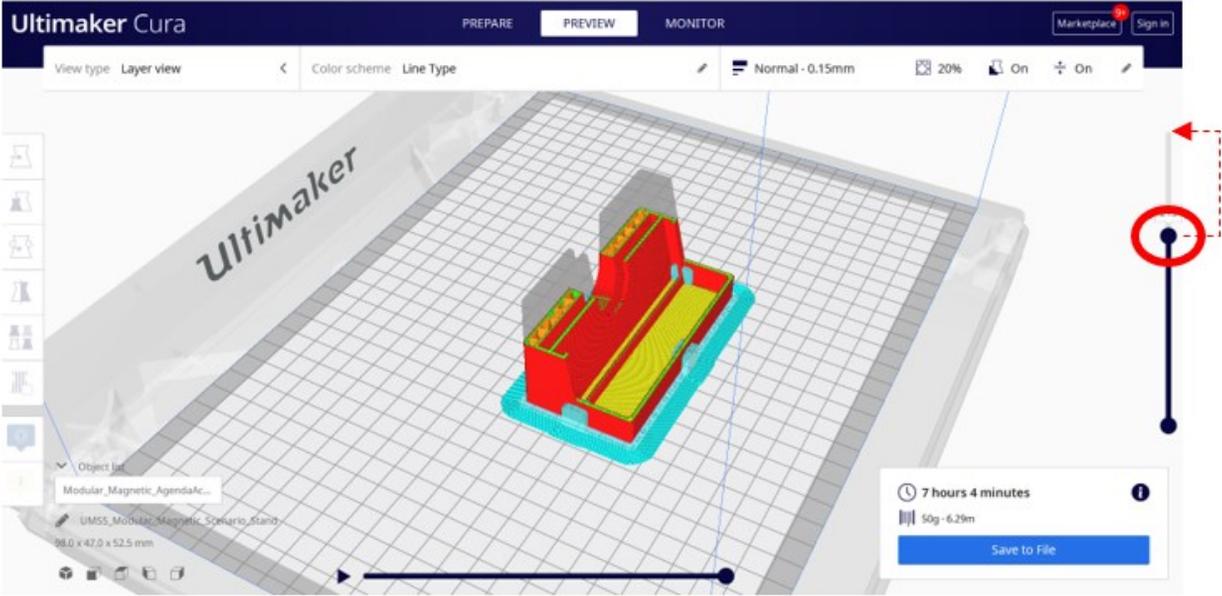
1. Click view type dropdown.

2. Select X-Ray View | Areas highlighted in red are issue areas that likely will prevent this model from successfully printing.

It is not recommended to try printing a model with errors shown in the X-Ray view. | |

25	Note the total number of grams and total print time.	On the bottom right after slicing, there will be a total amount of time to print and the number of grams of plastic used to complete the print job. You will need these details to reserve a 3D printer at the library.	<mark>You will want to make a reservation for a printer that is longer than the estimated print time (at least an additional 30 minutes).</mark> Cost of 3D printing is based on the number of grams of filament you use.	 **Cost to 3D Print** 25 cents per gram
26	Save file to USB drive.	Insert USB drive into computer. The save button will update with a new label. Click "**Save to Removable Drive**" This process saves a **.UFP** file to your USB drive.	You will plug this USB drive with your saved file into the 3D Printer to print your object. Flash drive must be formatted using FAT32 and have a total storage between 1 and 32 GB.	 FAT32 formatting required 1 – 32 GB Total Capacity

27	Reserve a 3D printer online	See the library website to book one of the Ultimaker S5 3D printers at the library. We recommend adding **at least** 30 minutes to your reservation for: • Check-in • Safety Review • Machine boot up • Machine cool down • Clean-up	It is not possible start a 3D print, pause and then resume 3D printing that same object at a later time. You will need a reservation that is long enough to include additional time to check-in, review the safety precautions, and clean up time.	friscolibrary.com/3d-printing
28	Bring to the library: Library Card Photo ID USB Drive	You will need your photo ID at check in. Be sure you are using your own library card account. All fees are billed to the library card account used to make the reservation.	Because fees are billed to the library card account making the reservation, we are unable to allow guests to use another individual's card to book time on a 3D printer.	