

HOW TO 3D PRINT WITH THE ULTIMAKER S5 USING THE CURA SOFTWARE



HOW TO 3D PRINT WITH THE ULTIMAKER S5 USING THE CURA SOFTWARE

Standard Prints & PVA Supports



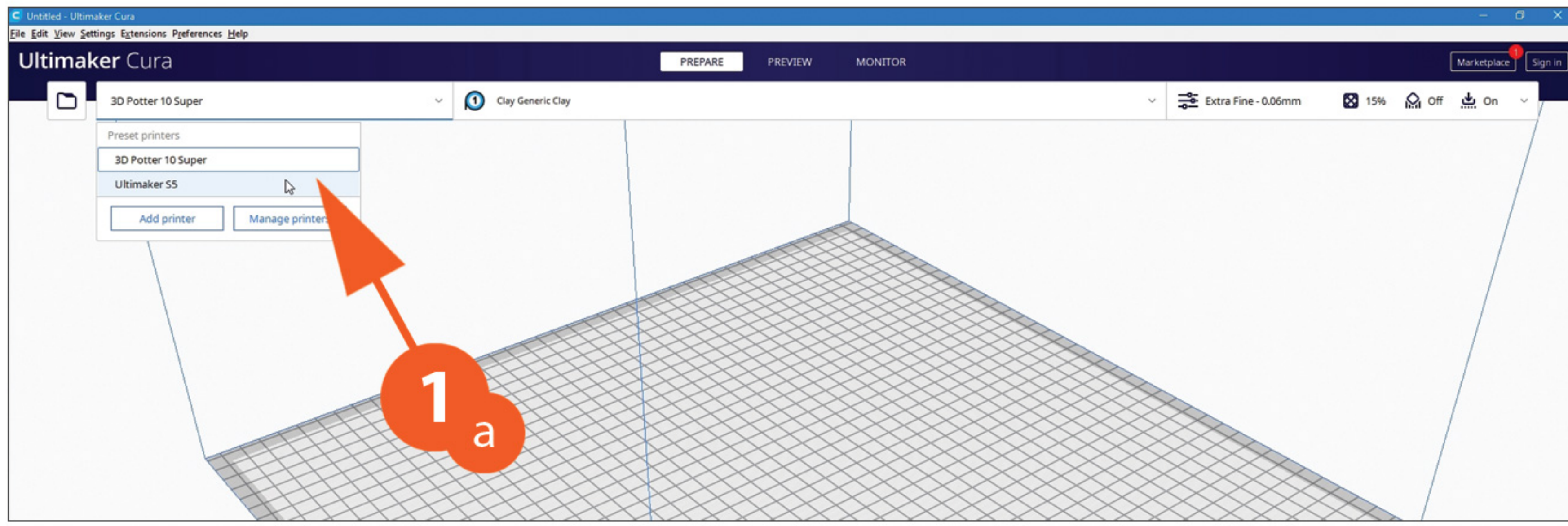
Version v1

Published December 29, 2022

Author Don Gillard

Langara College Makerspace

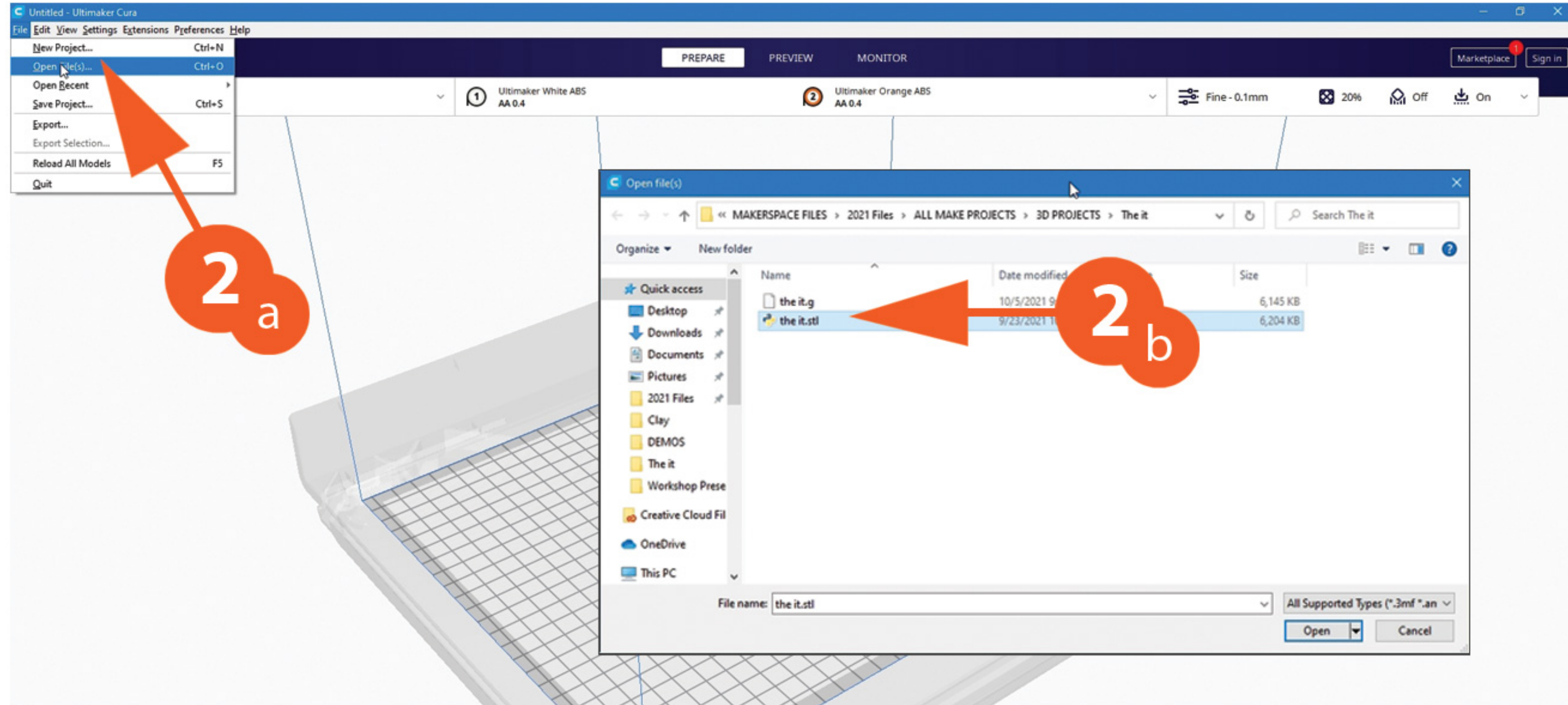
All rights reserved, Langara College 2022-23



1

Open Cura.

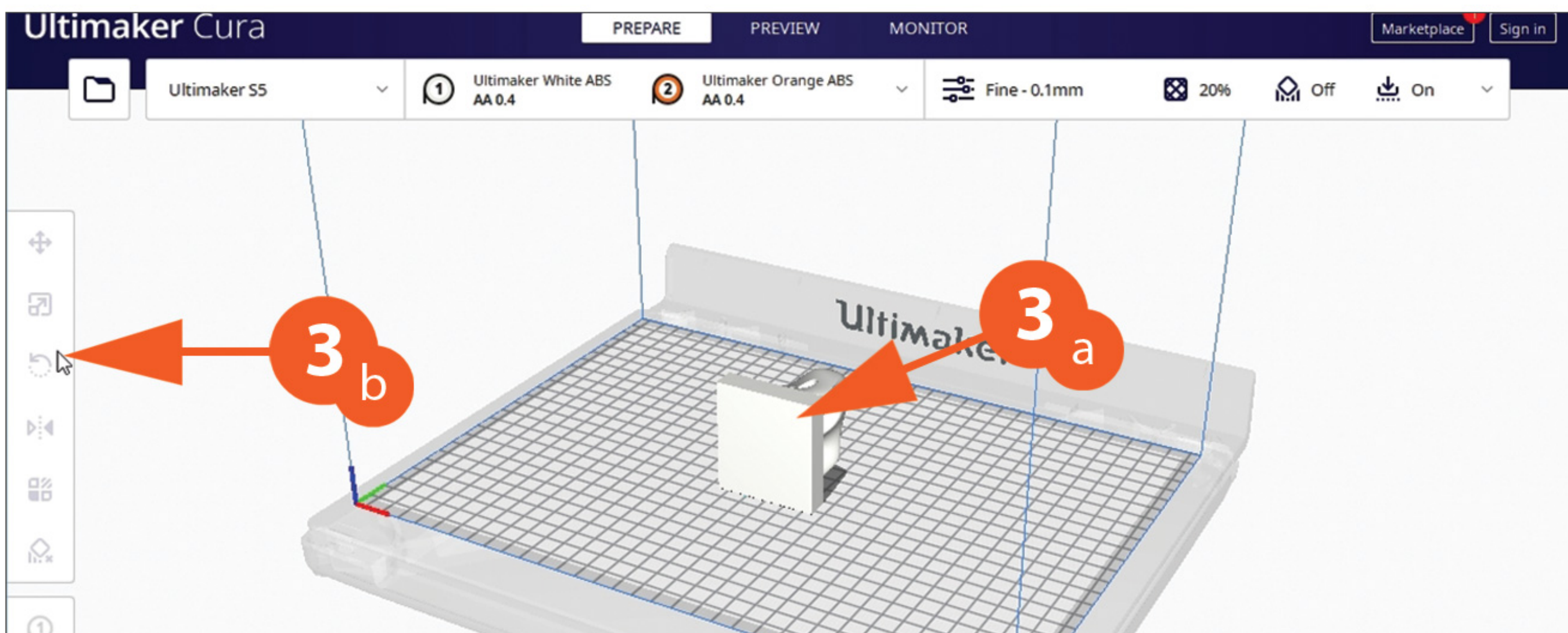
1a - Set Profile to the Ultimaker S5 printer.



2

2a - Open file.

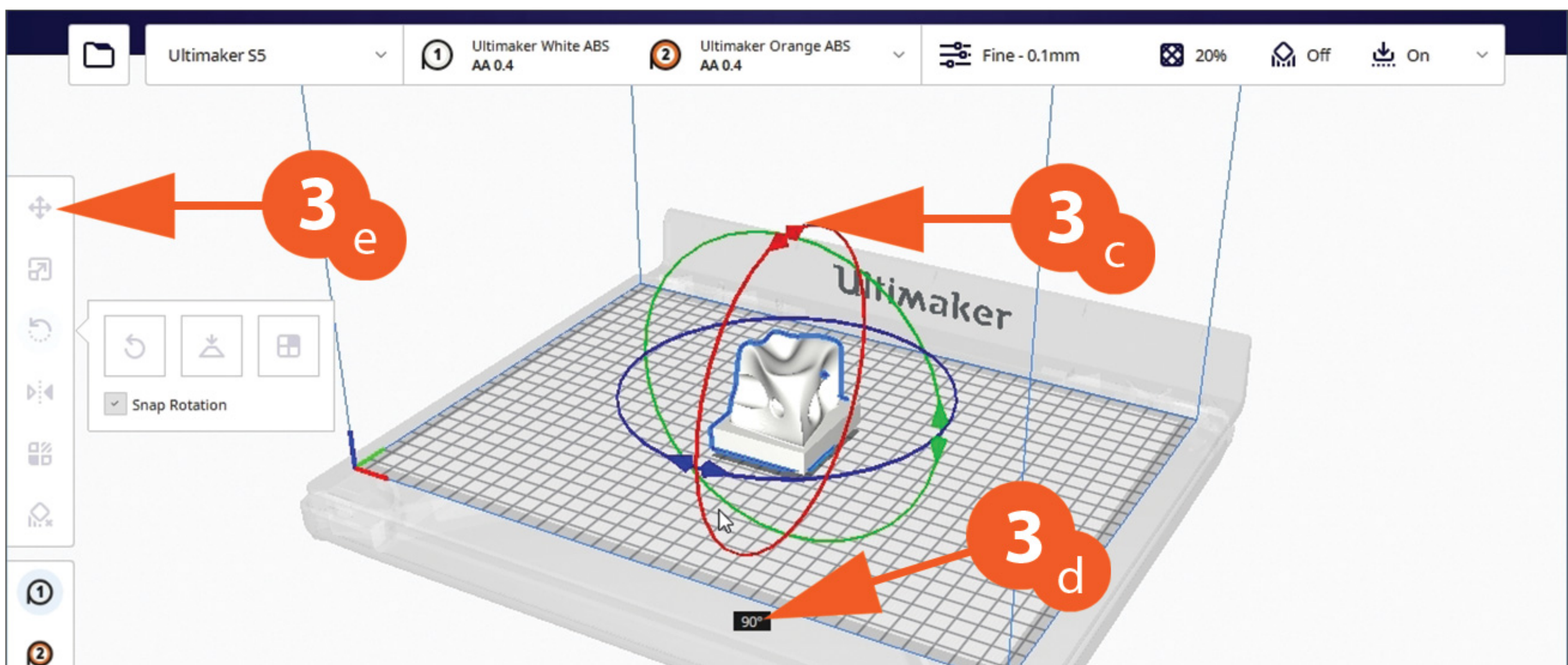
2b - Select your; STL, OBJ or 3MF File.



3

3a - Select your object.

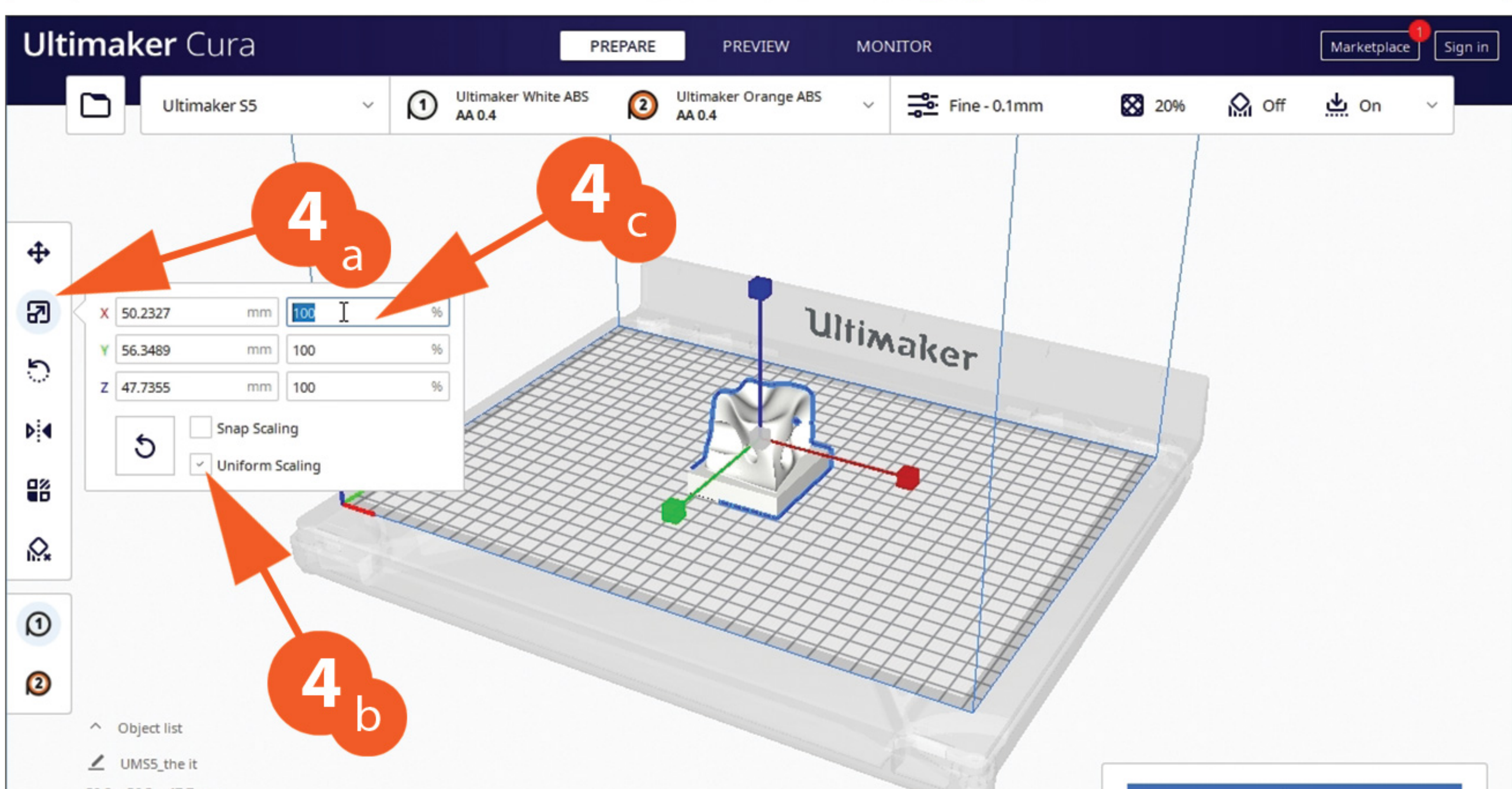
3b - Select the Rotate icon, if you want to change the orientation.



3c - Choose 1 of the rings to rotate the object.

3d - Rotate to the desired position.

3e - If you want to move your object, Select the Move icon, & select your object and move it.



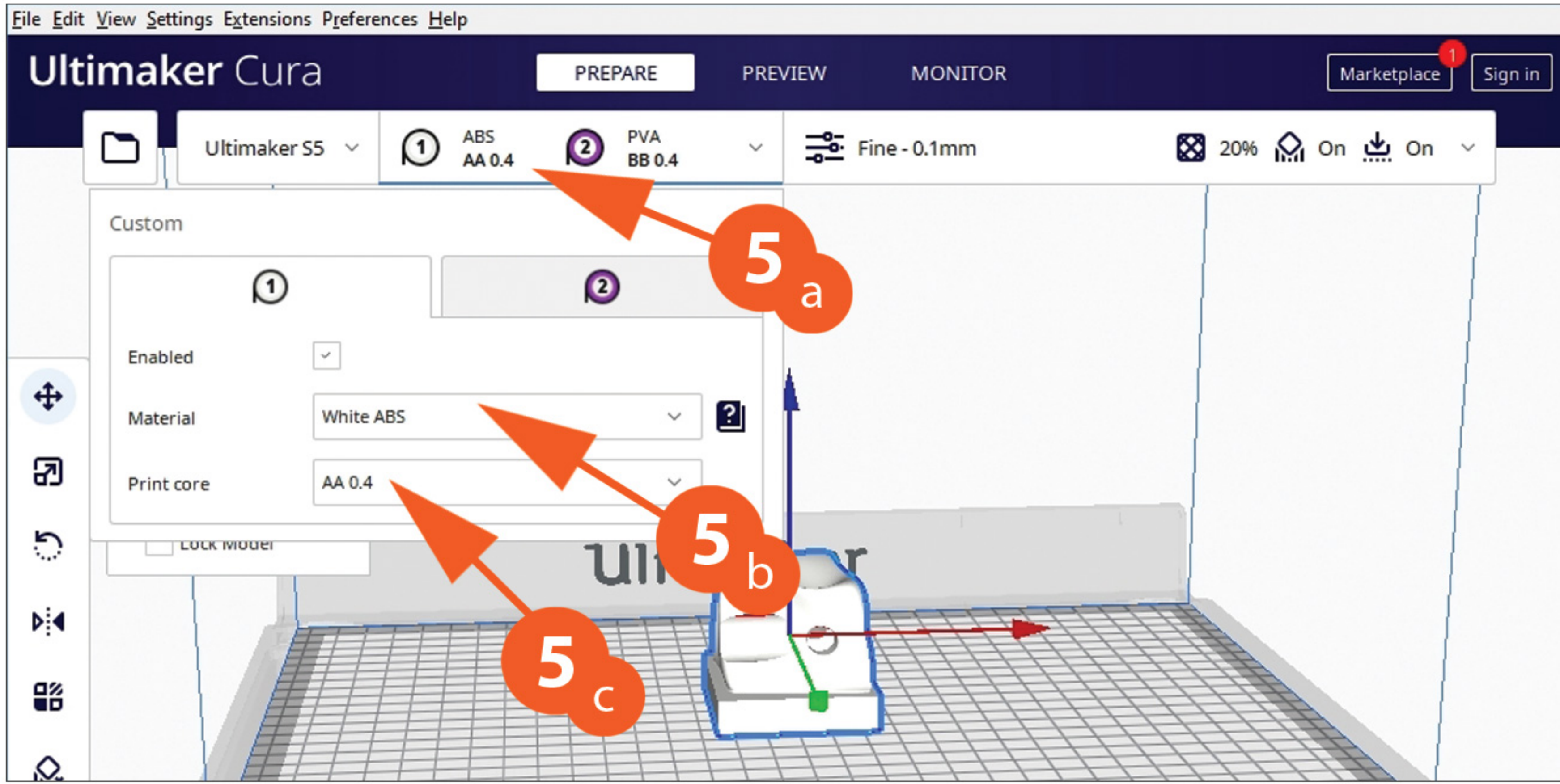
4

4a - Select the **Scale** icon, if you want to scale up or down your object.

4b - Make sure to have **Uniform Scaling** checked or you will change the proportions of your object.

4c - Enter the new scale number in the X - % box (it is default at 100%).

Note:
 STL file format is in millimeters (mm), so your model should be built in mm.

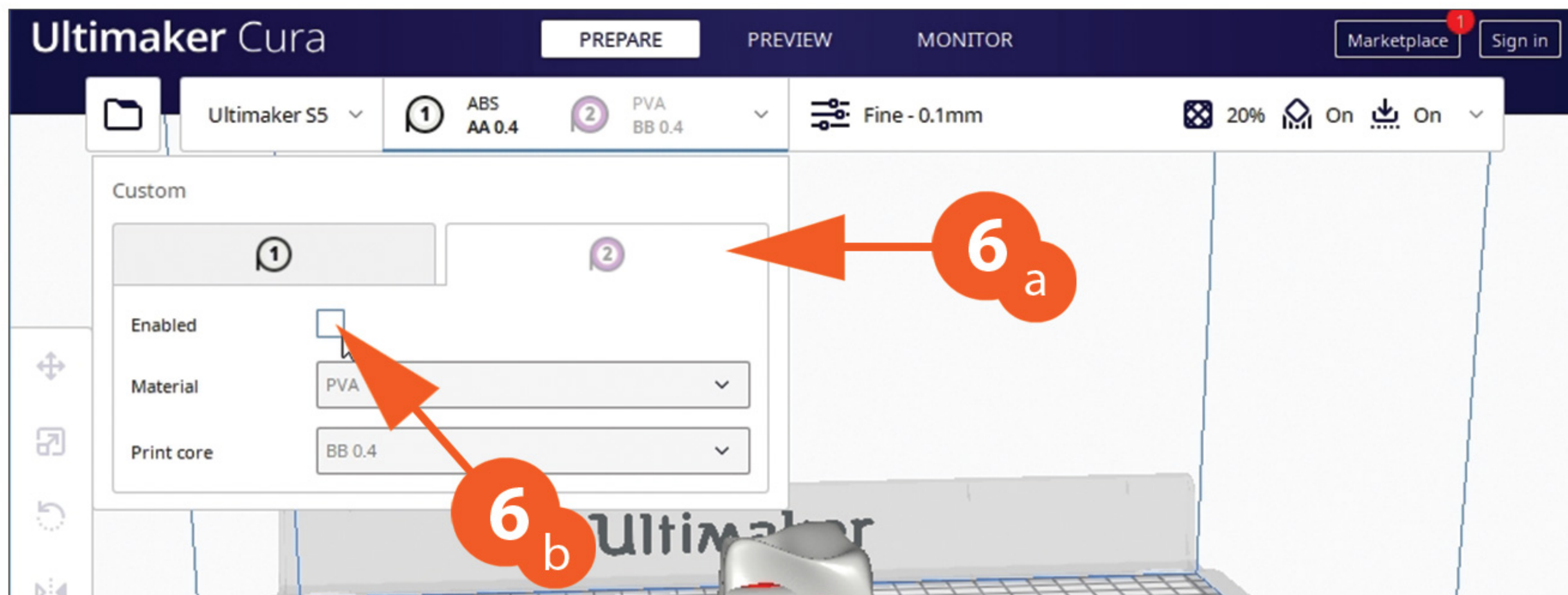


5

5a - Click on the **Material Menu** bar.

5b - Click **Enabled** & Select the Material you are using for material #1 (We use Ultimaker ABS colours, for most applications on the S5).

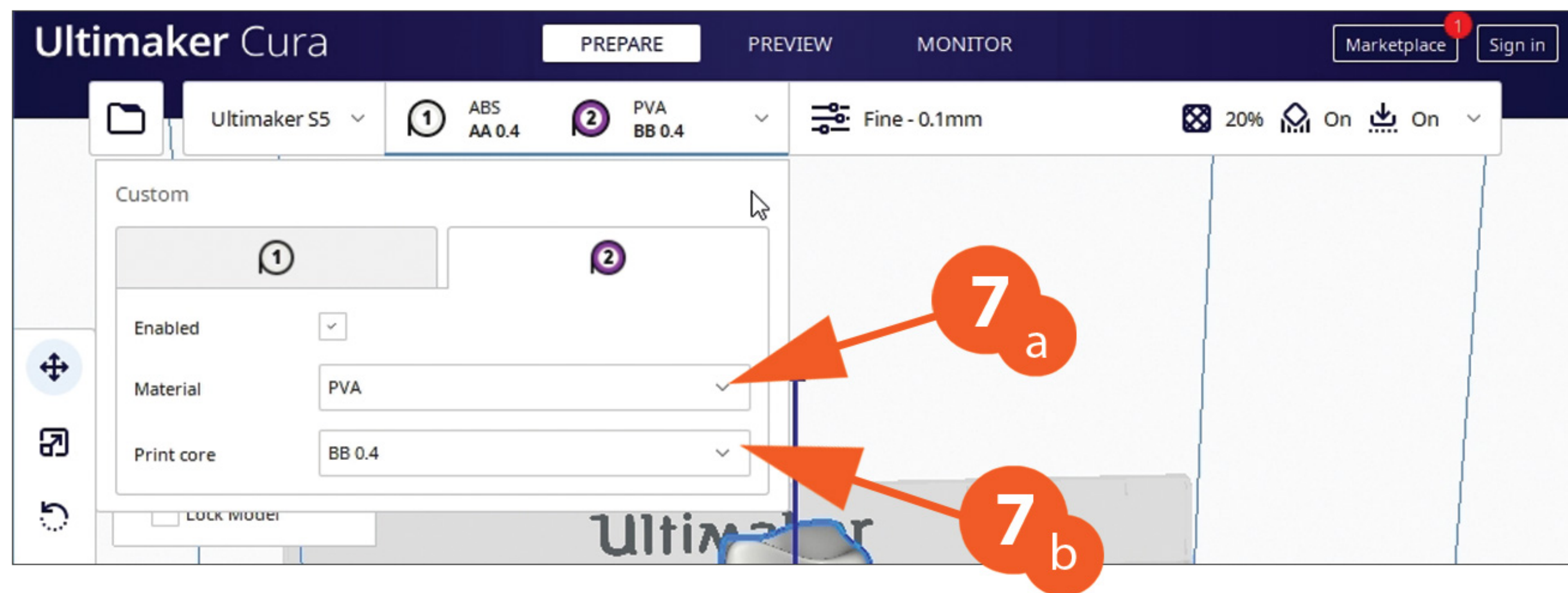
5c - The print core for ABS should be AA 0.4



6

6a - Select the second tab, Material 2.

6b - If you are not using a second material uncheck the **Enable** box.

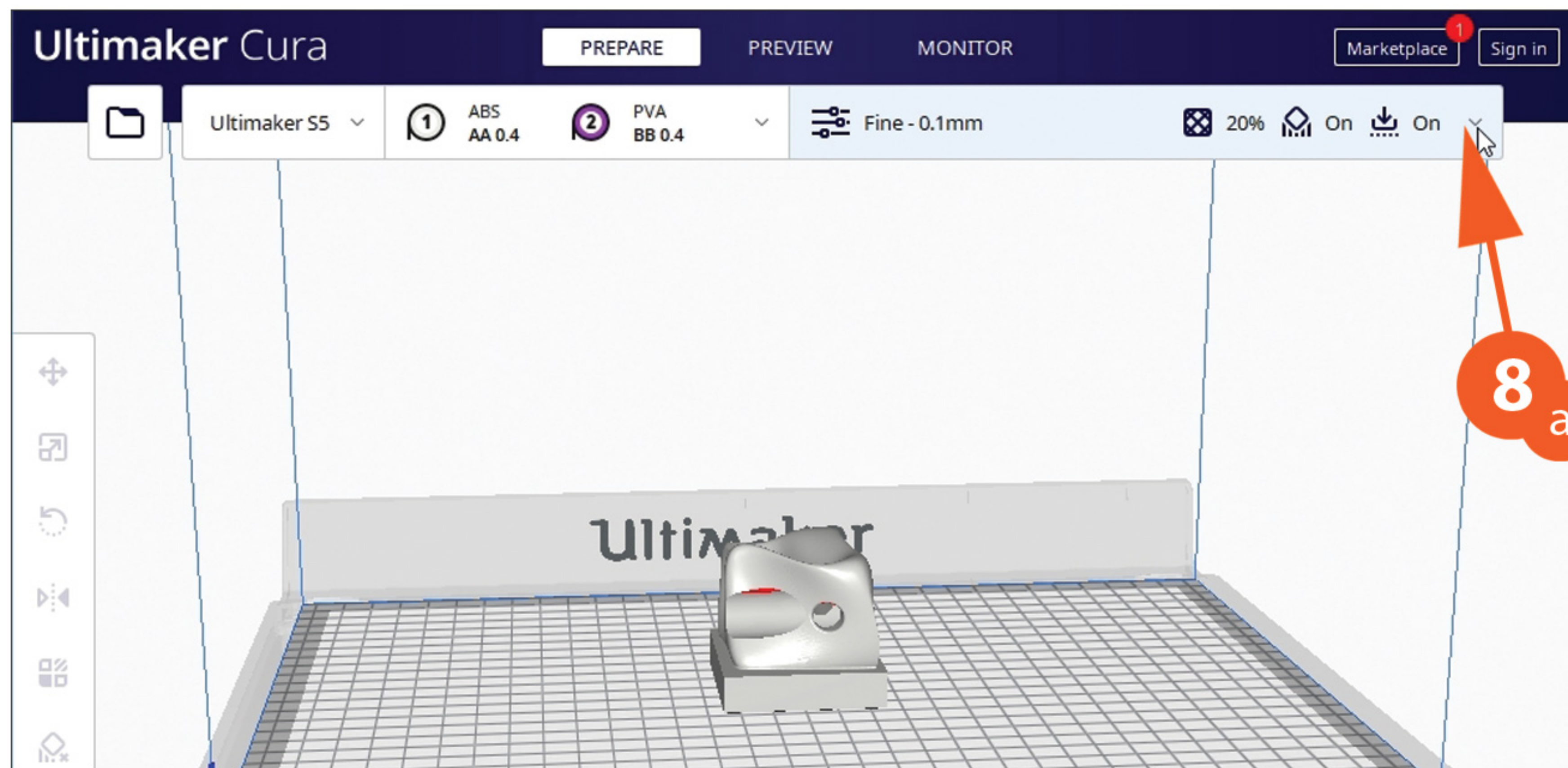


7

7 - If you want a dissolving support, you will need to use PVA.

7a - Select **Generic PVA**.

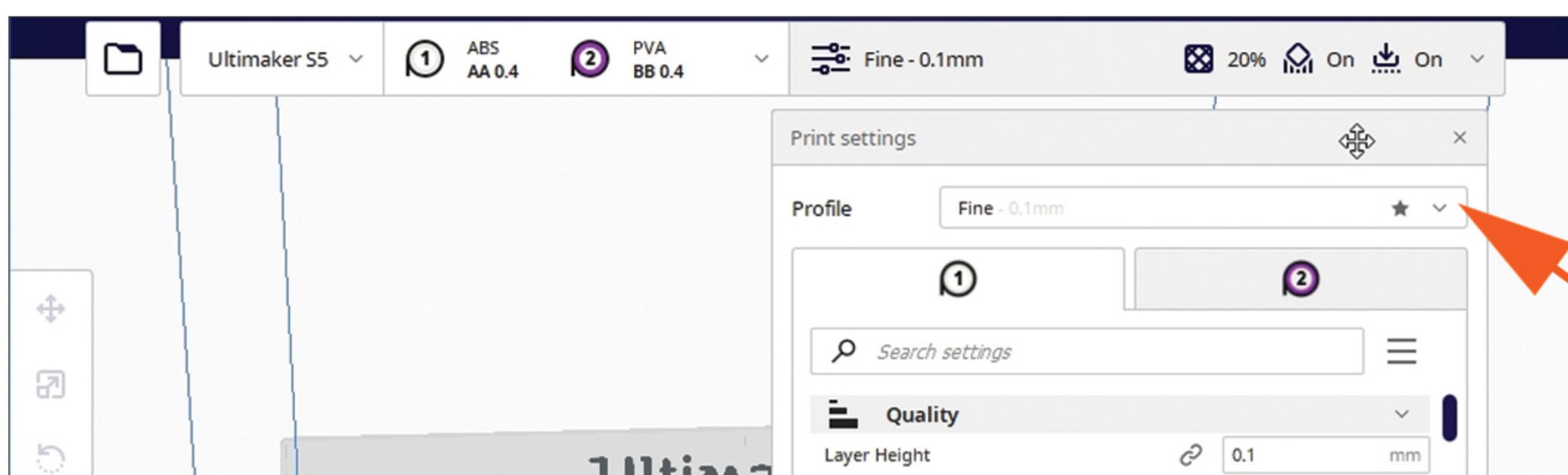
7b - Select the BB 0.4 print head for PVA. Then ask the tech to change the print head on the S5 printer for you.



8

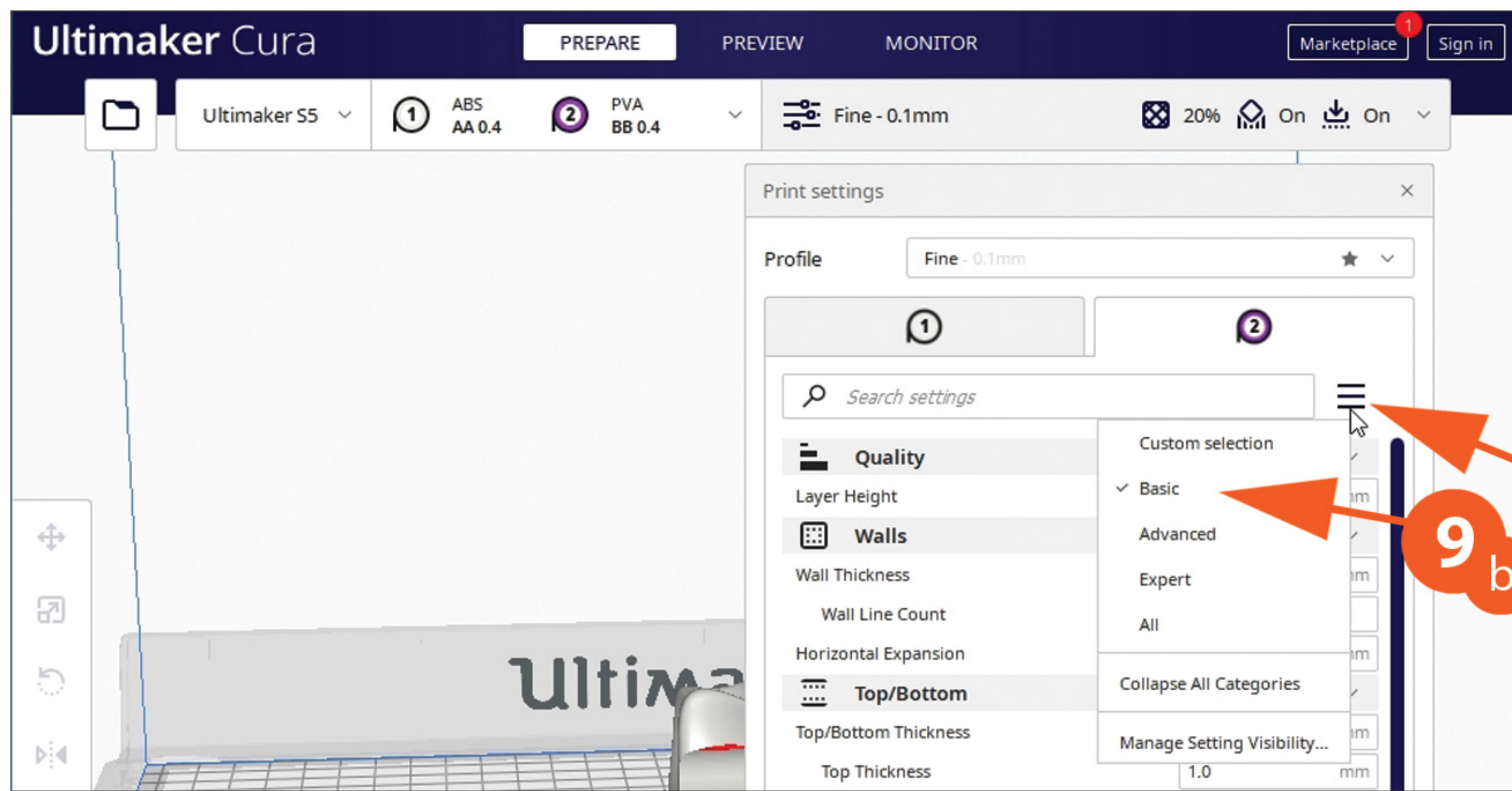
8a - Click on the **Settings Menu** bar to open up the settings menu.

8b - Select one of the Default Setting:
 - Extra Fine - .06mm resolution (layers)
 - Fine - .1mm resolution (layers)
 - Normal - .15mm resolution (layers)
 - Fast - .2mm resolution (layers)



8

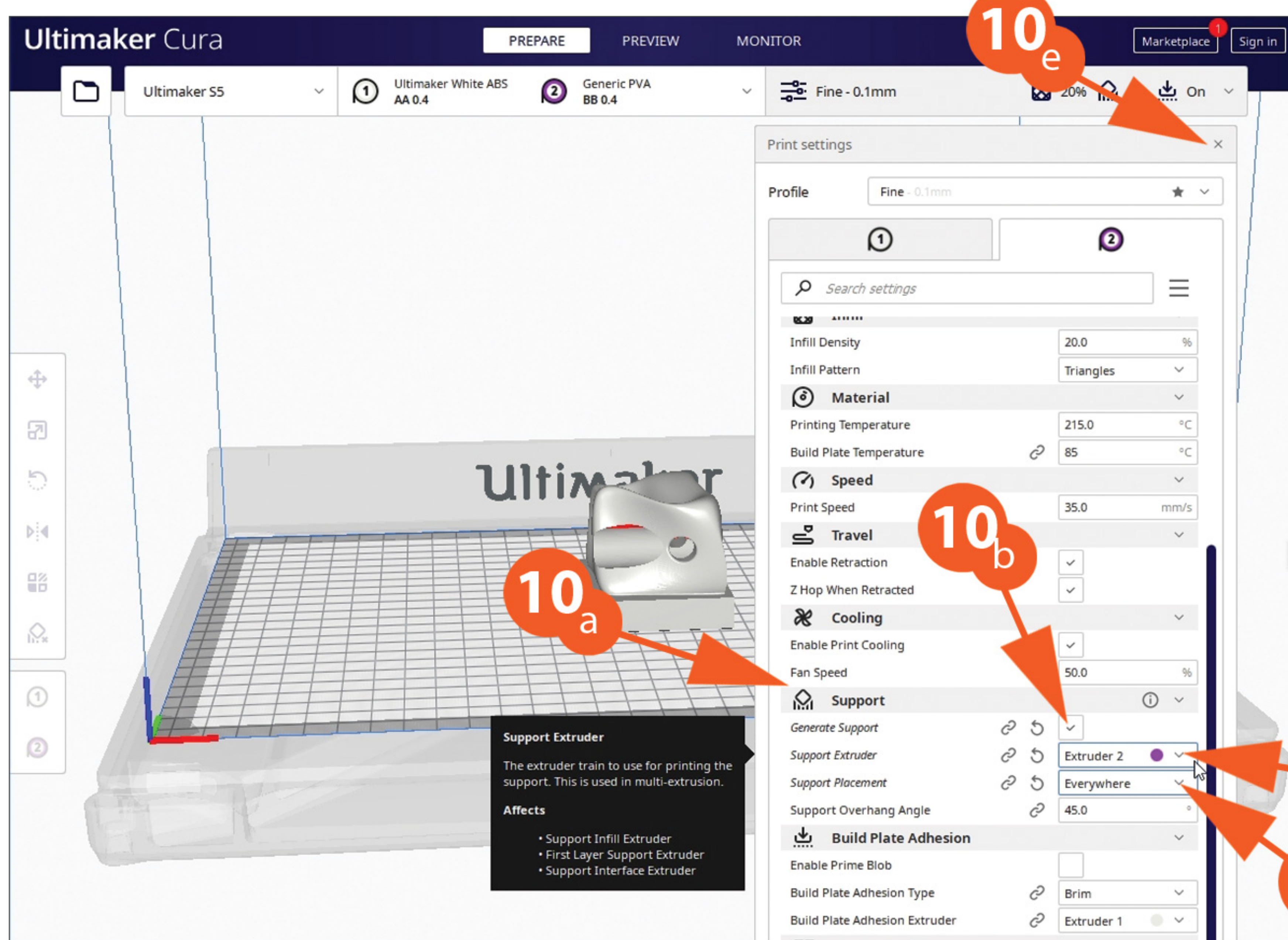
When the pop-up appears, click on **Keep Changes**.



9

9a - Click on the **Hamburger** (3 lines).

9b - Select **Basic**.



10

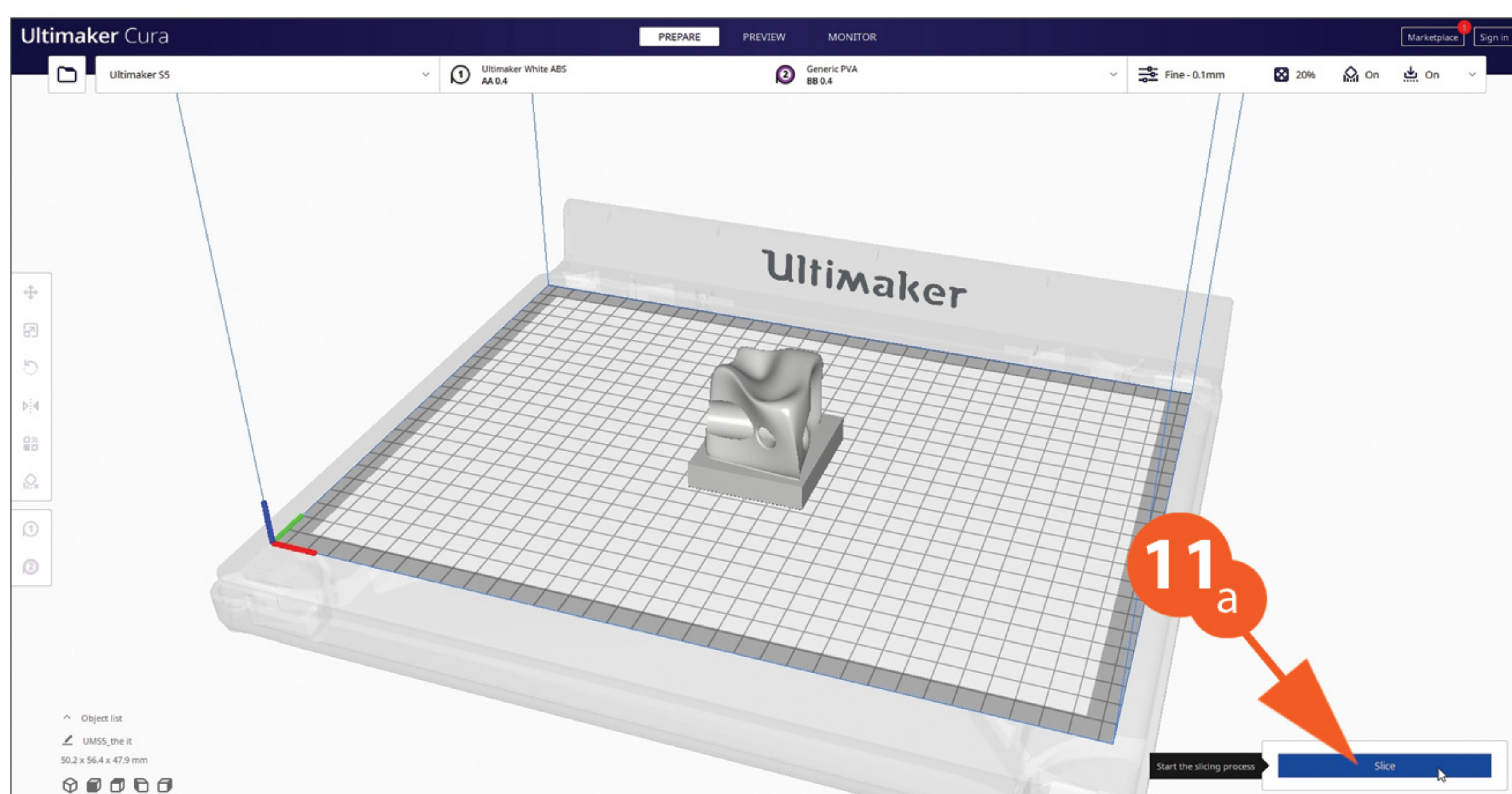
10a - Scroll down to **Support**.

10b - If you are going to use a support, check the **Generate Support** box if not leave it unchecked.

10c - On the **Support Extruder**, make sure **Extruder 2** is selected.

10d - For **Support Placement**, select **Everywhere**.

10e - Close the Menu.



11

11a - Click on the **Slice Button** to slice your model for the printer.

11b - You will now see the print time...

11c - As well as the grams of filament required.

11d - Click on **Save To Disc**, and save it to a USB stick.

11e - To Preview the print, click on **Preview** You can use the sliders to see the construction of your print in different stages.



HOW TO 3D PRINT WITH THE ULTIMAKER S5 USING THE CURA SOFTWARE

Two Colour Prints



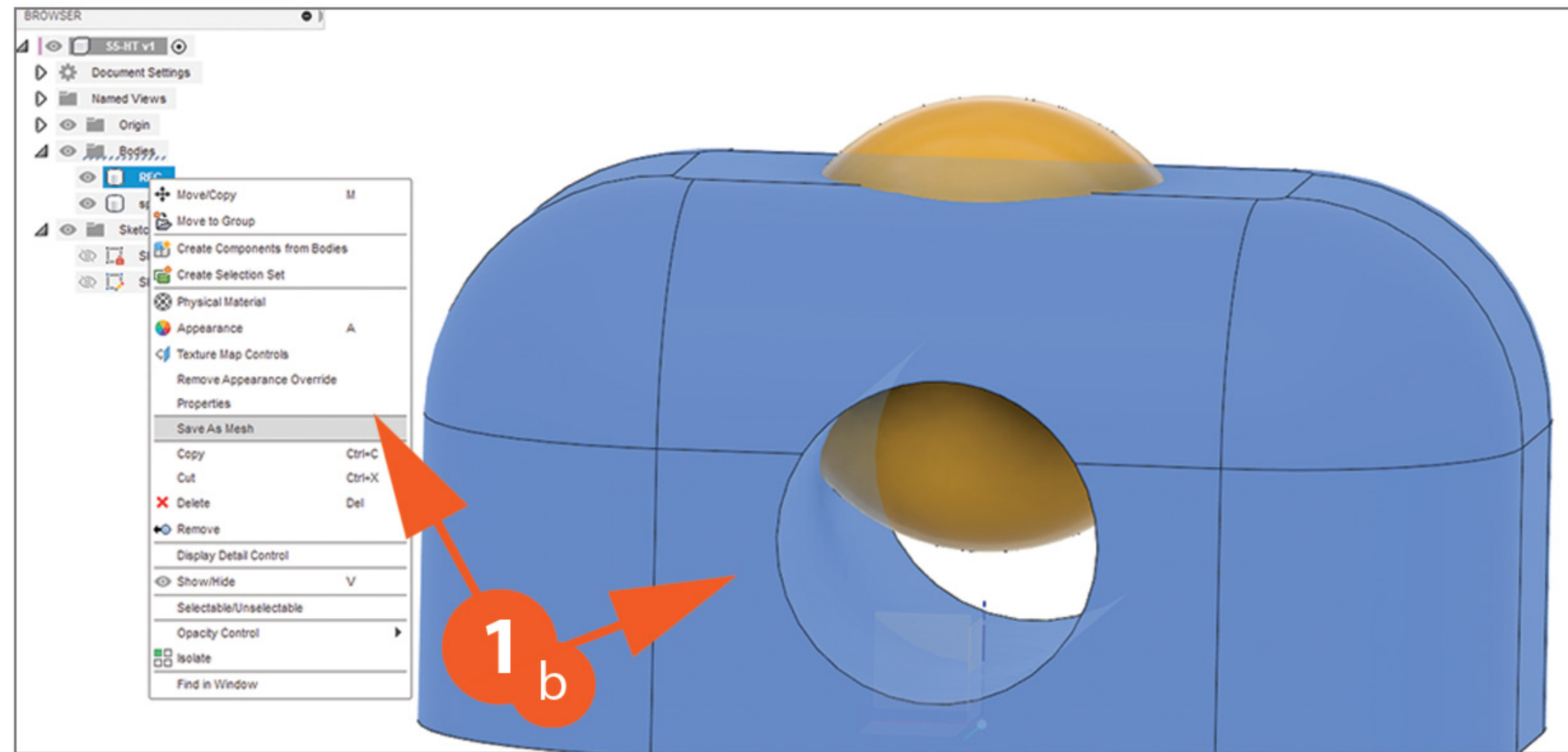
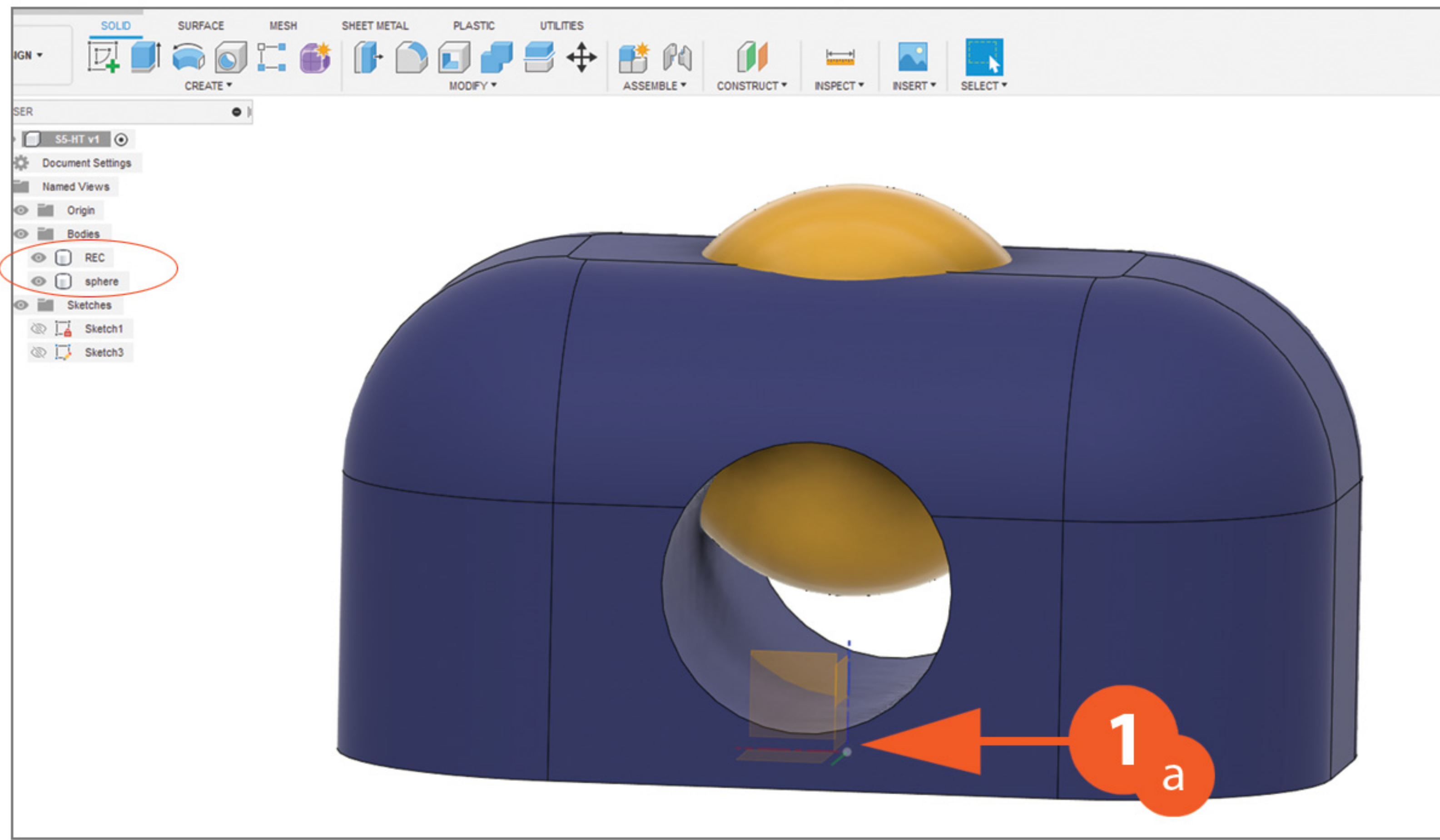
Version v1

Published December 29, 2022

Author Don Gillard

Langara College Makerspace

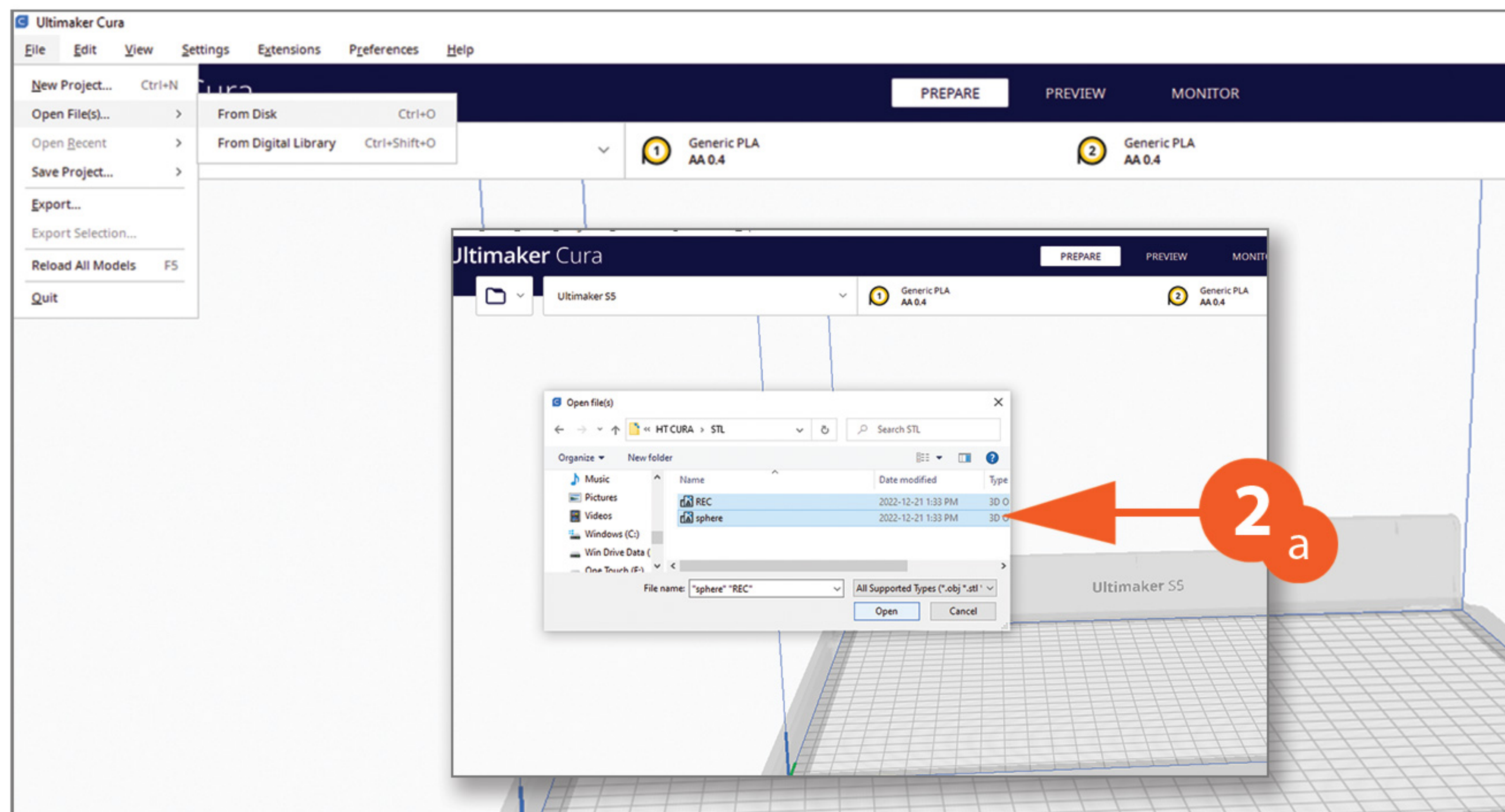
All rights reserved, Langara College 2022-23



1

1a - In Your 3D Creation program , such as Fusion 360, Rhino 3D and Solidworks you need to build your 2 colour objects using the same origin.

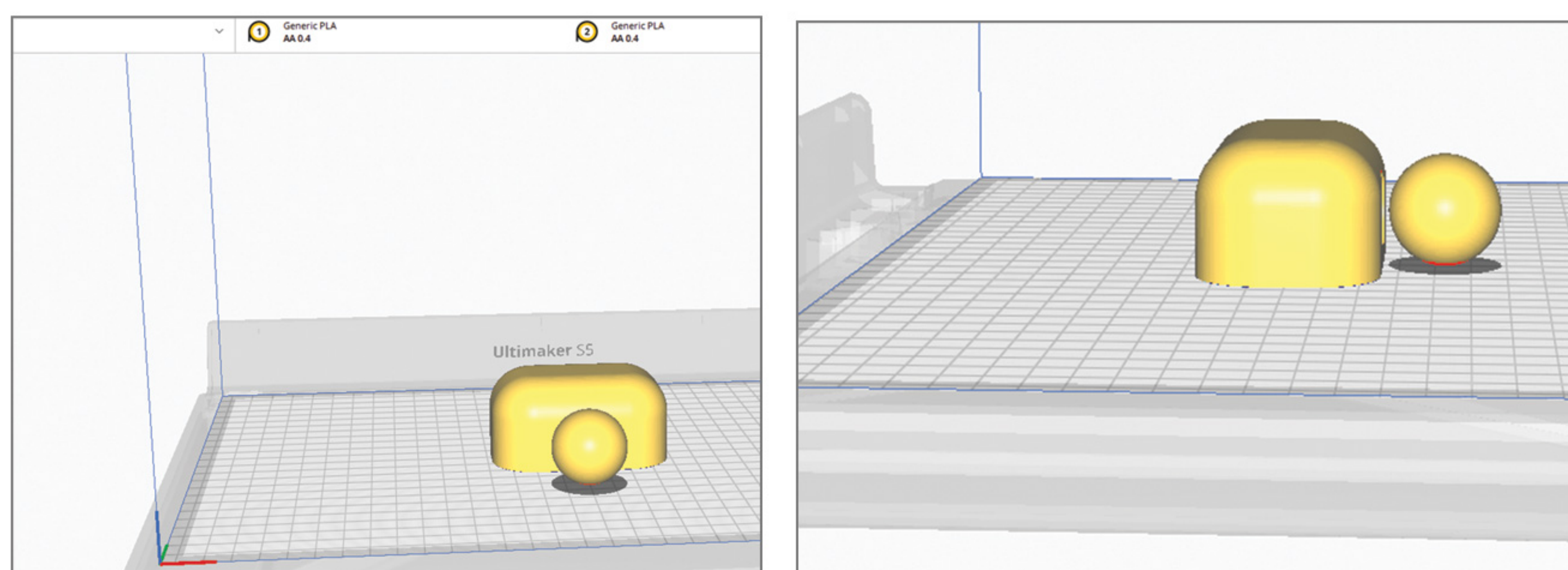
1b - Select one of the objects, and save it as an STL. Repeat with the second object.



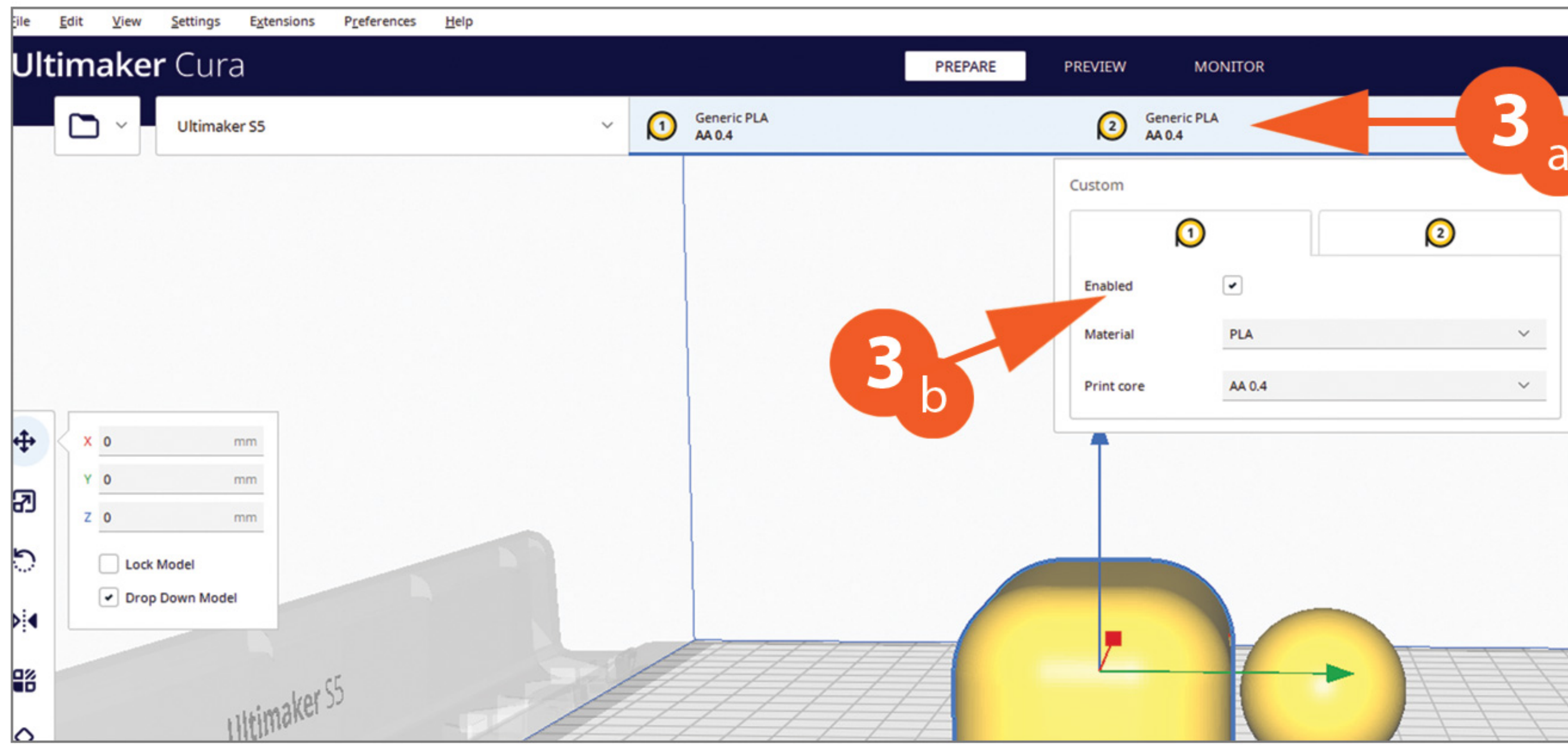
2

2 - Open Cura

2a - Holding down the shift key, select and open your two STL files.



You will see your two objects on the print bed and separated.



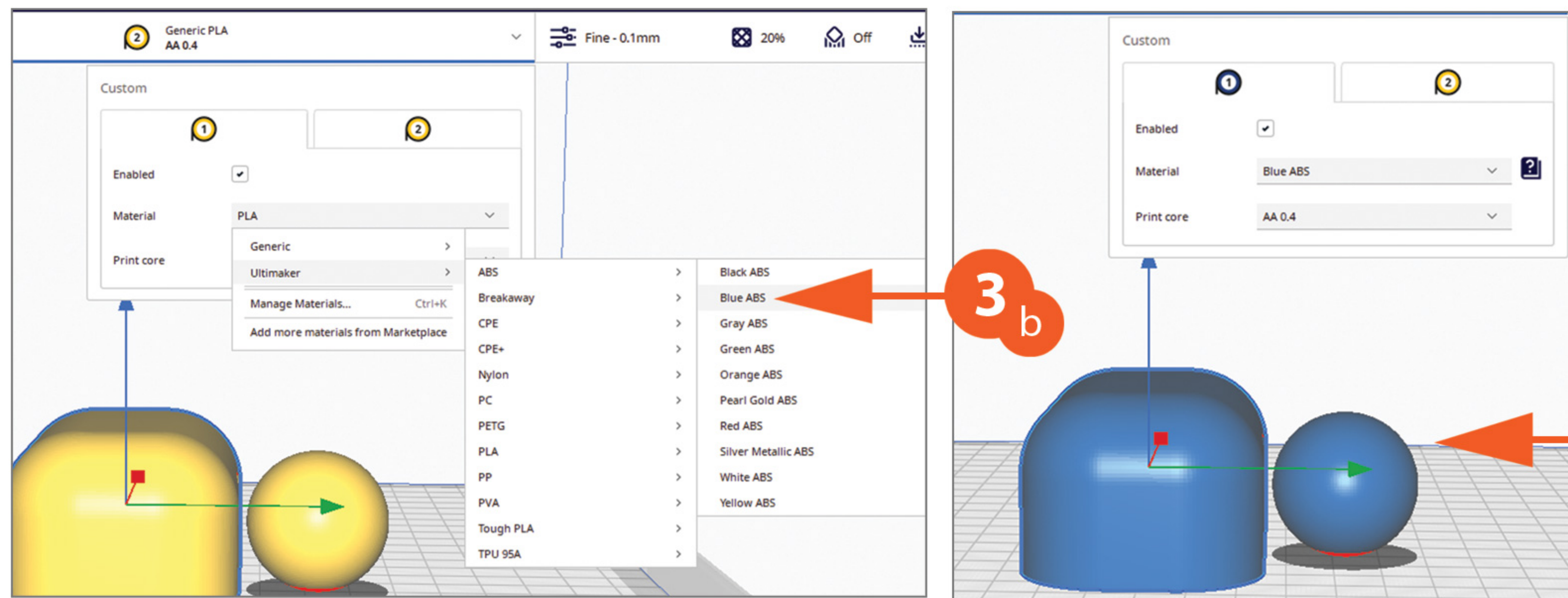
3

3a - Select the **Materials Panel**.

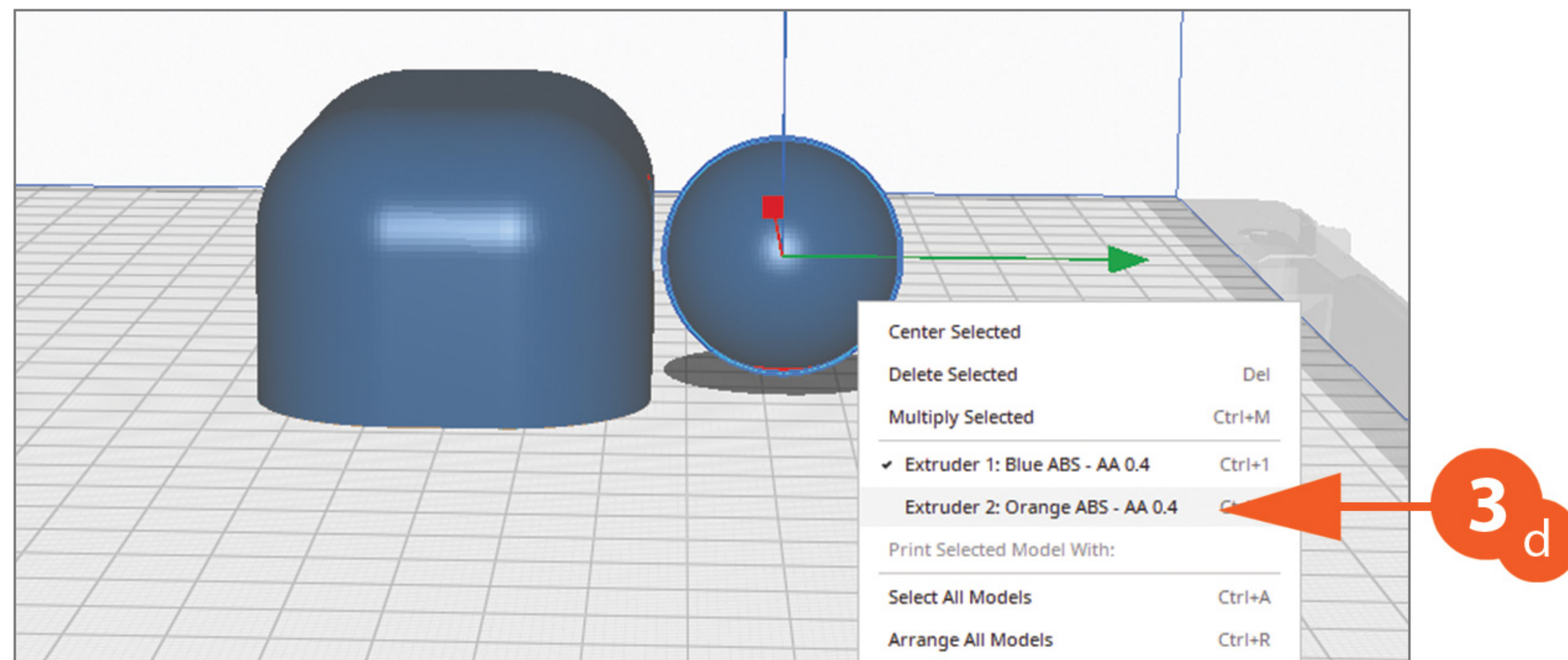
3b - On tab one, select your material.

3c - Both objects will take on the new colour.

Select the second tab, and select its material.



3d - Now **Right Click** on the object that requires the second tabs colour/material, and select **Extruder 2**.

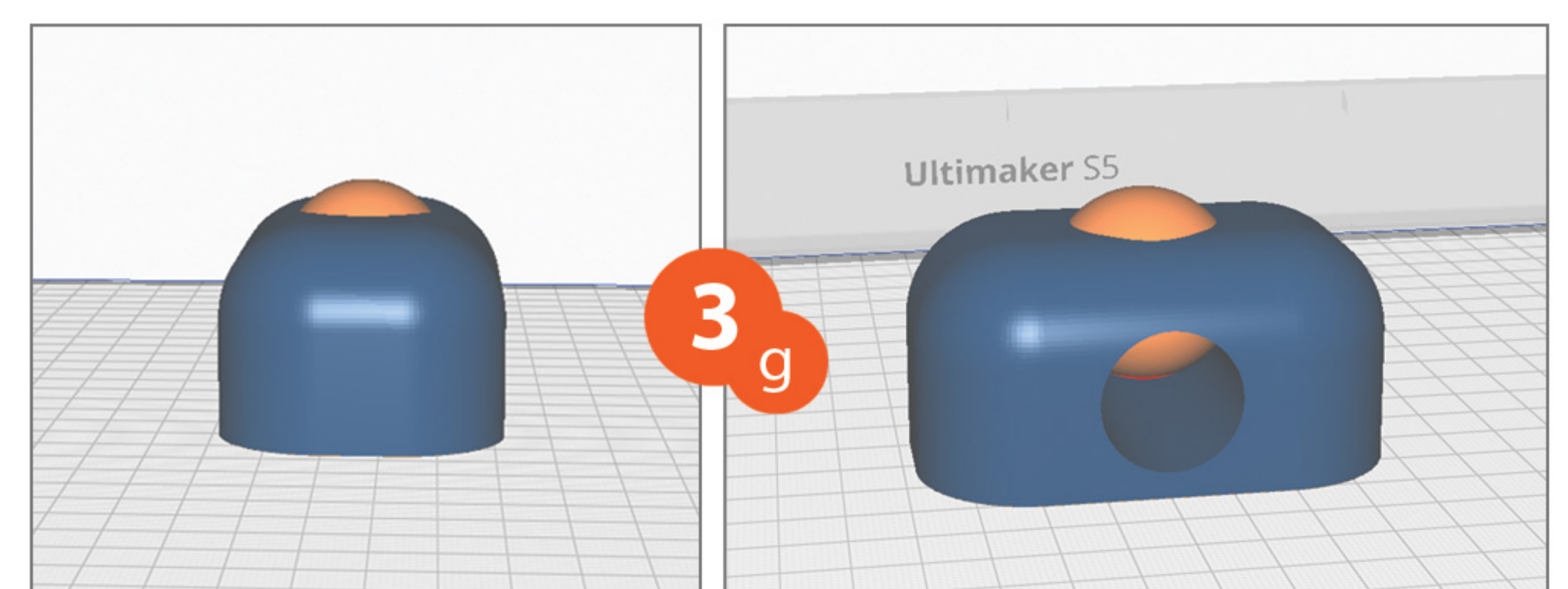
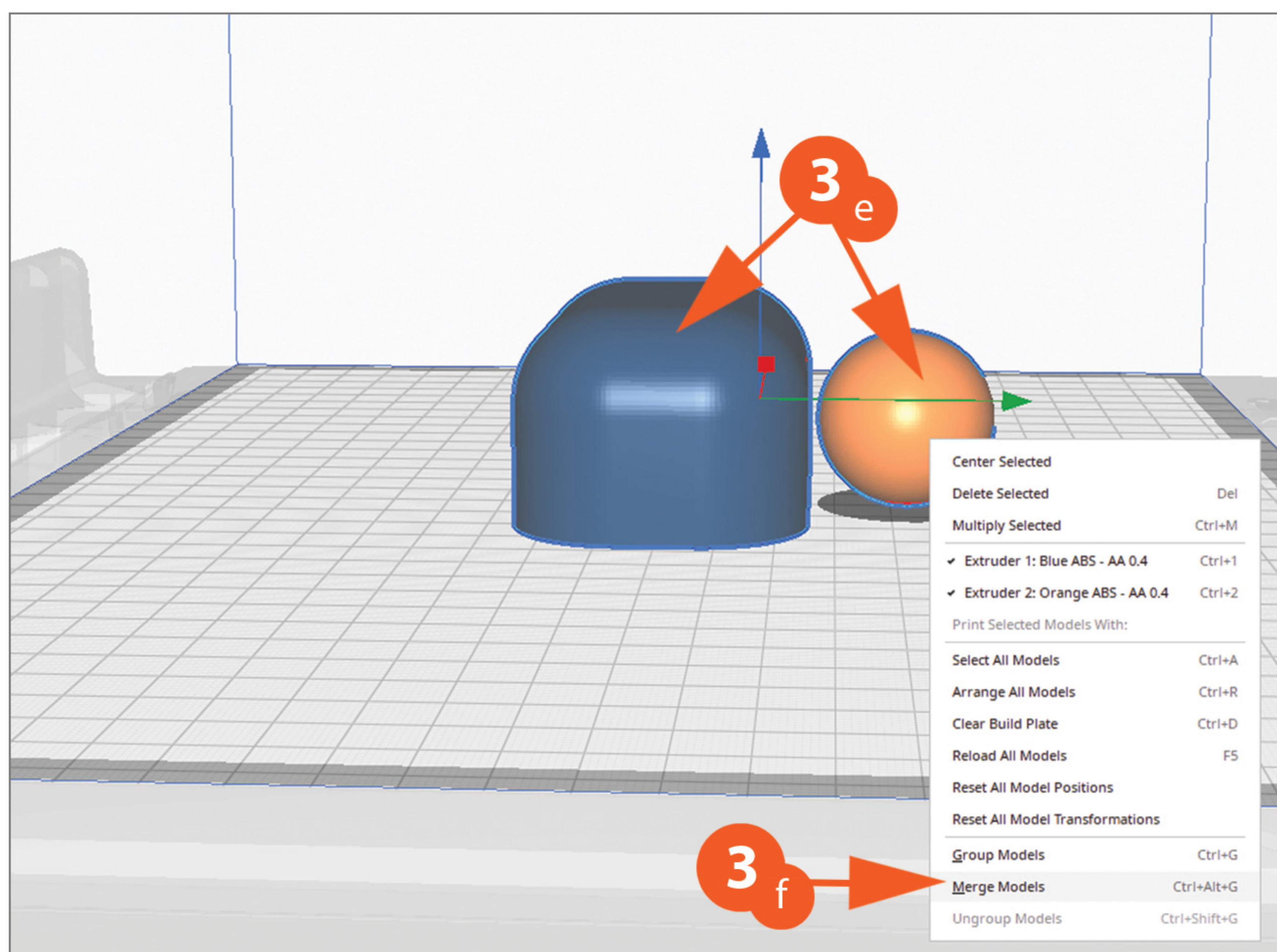


3e - Now Select both objects (use SHIFT key) and **Right Click**.

3f - Click on **Merge Models**.

3g - Models will now be together as per your 3D program.

Proceed to your Print settings, Slice & Print.



HOW TO 3D PRINT WITH THE ULTIMAKER S5 USING THE CURA SOFTWARE

Multiple Density Prints



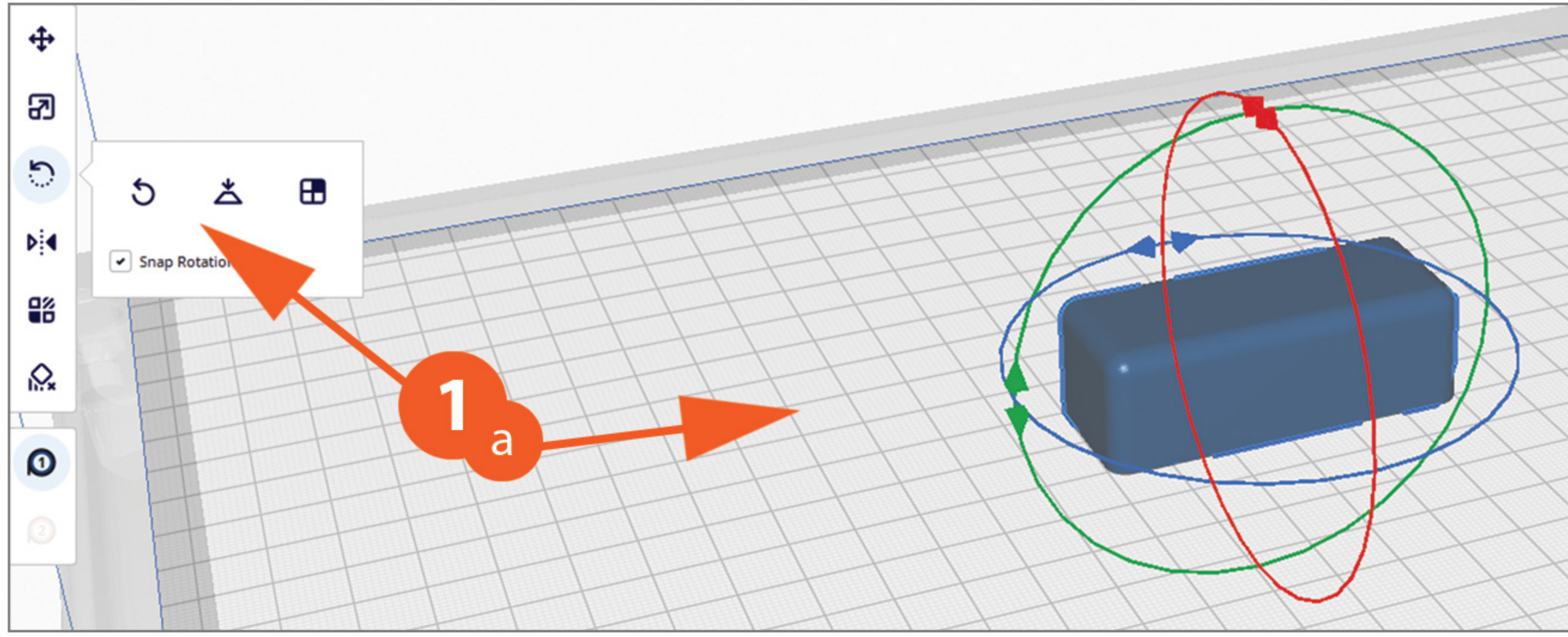
Version v1

Published December 29, 2022

Author Don Gillard

Langara College Makerspace

All rights reserved, Langara College 2022-23



1

Multiple densities are used where an area needs a more solid fill but not the entire object. When using plastic insert nuts, you would want a solid fill around the hole created for the nut. The following example is for solid fill around holes for insert nuts.

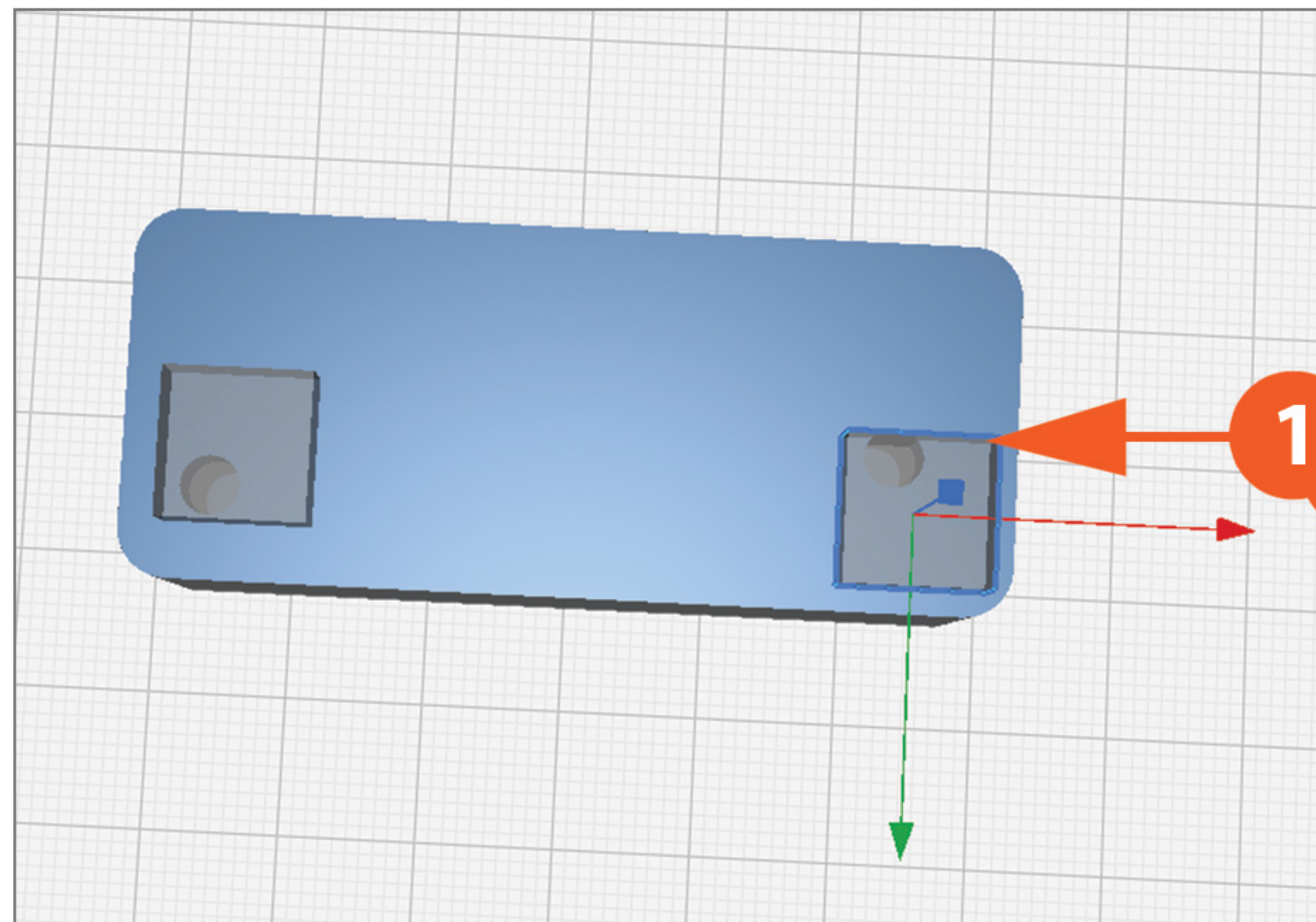
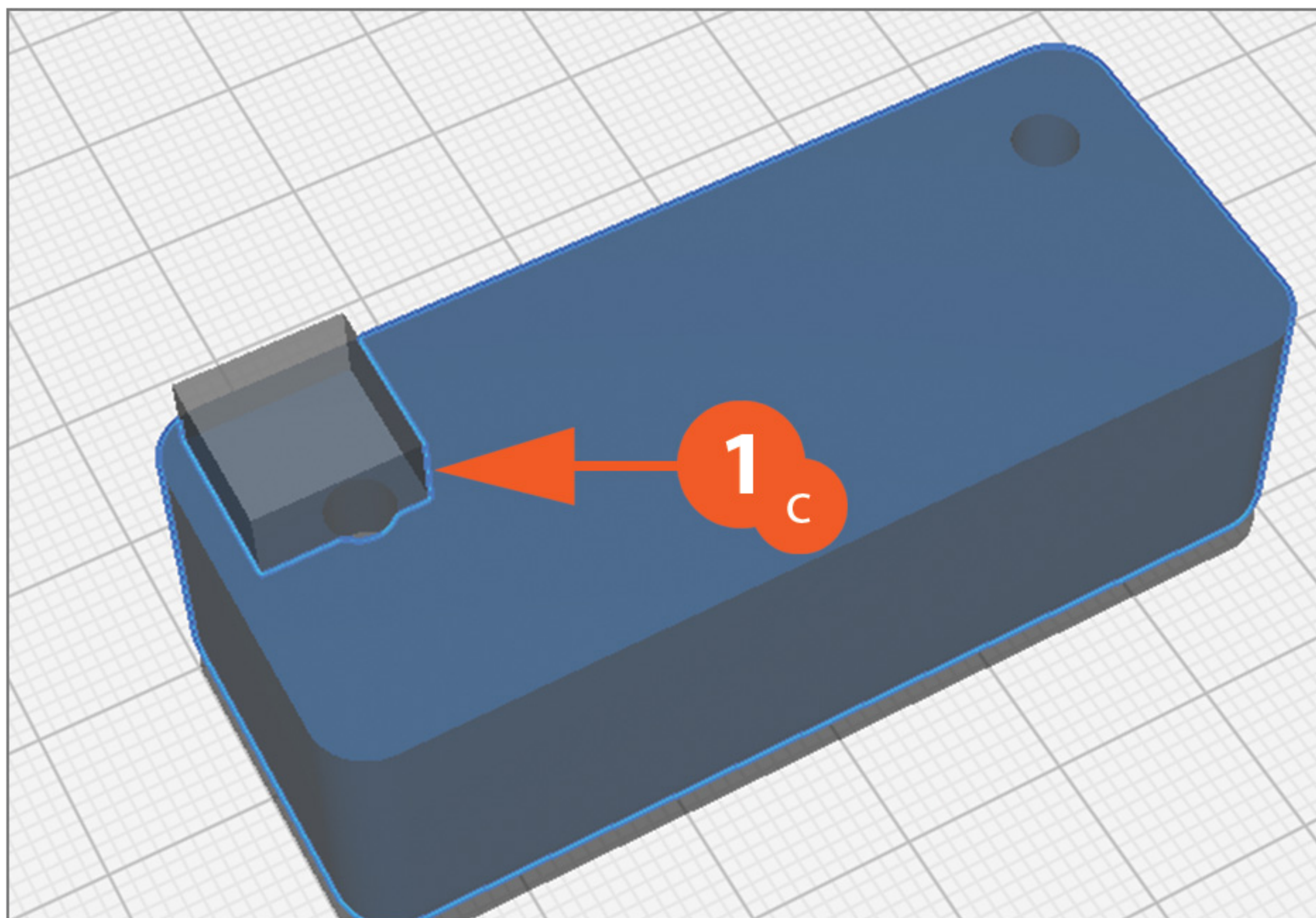
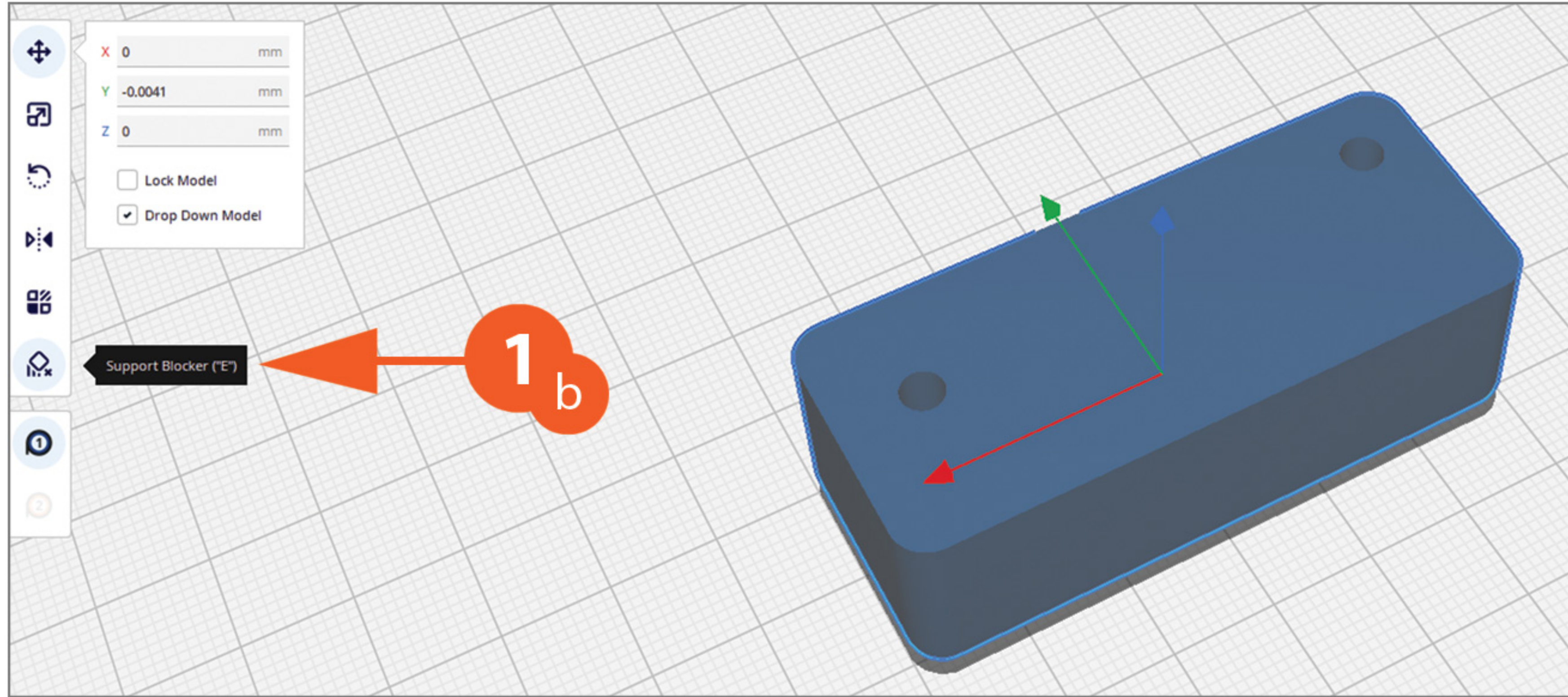
1 - Open your STL object in Cura.

1a - If you cannot see your holes, rotate your object so they are visible.

1b - With your object selected, Select the **Support Blocker Icon**

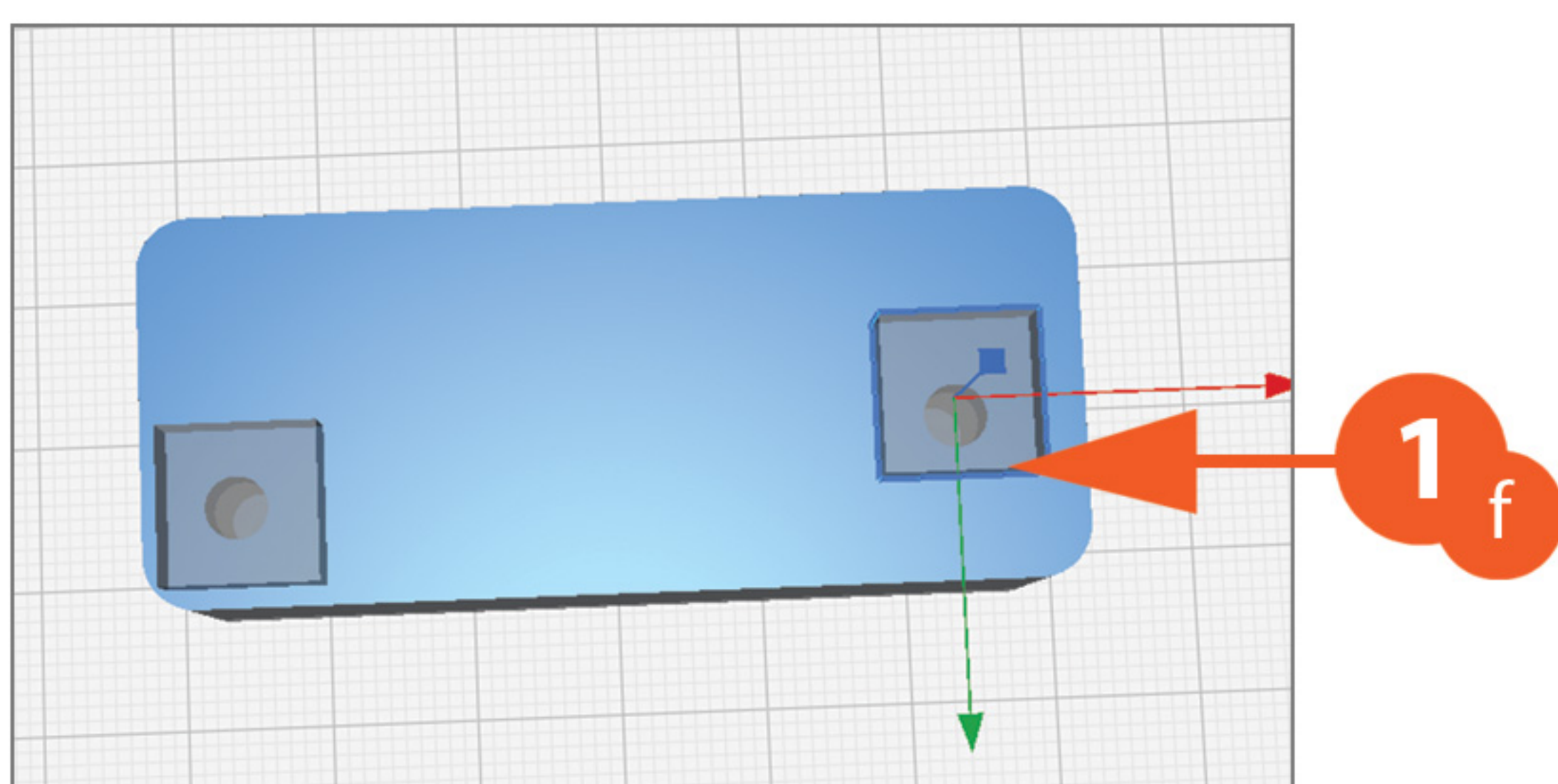
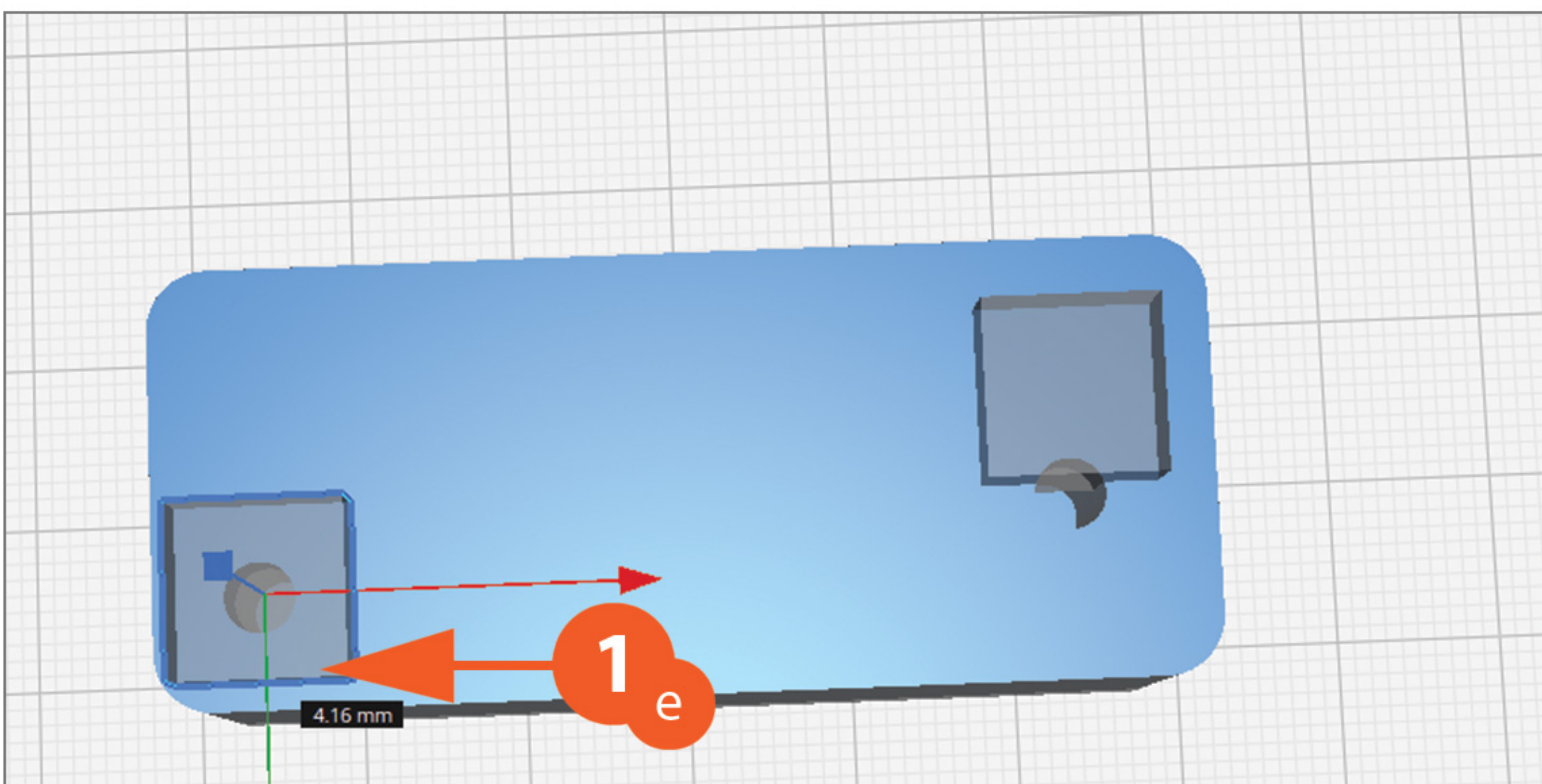
1c - Click on your object near the first hole. A semi transparent block will appear.

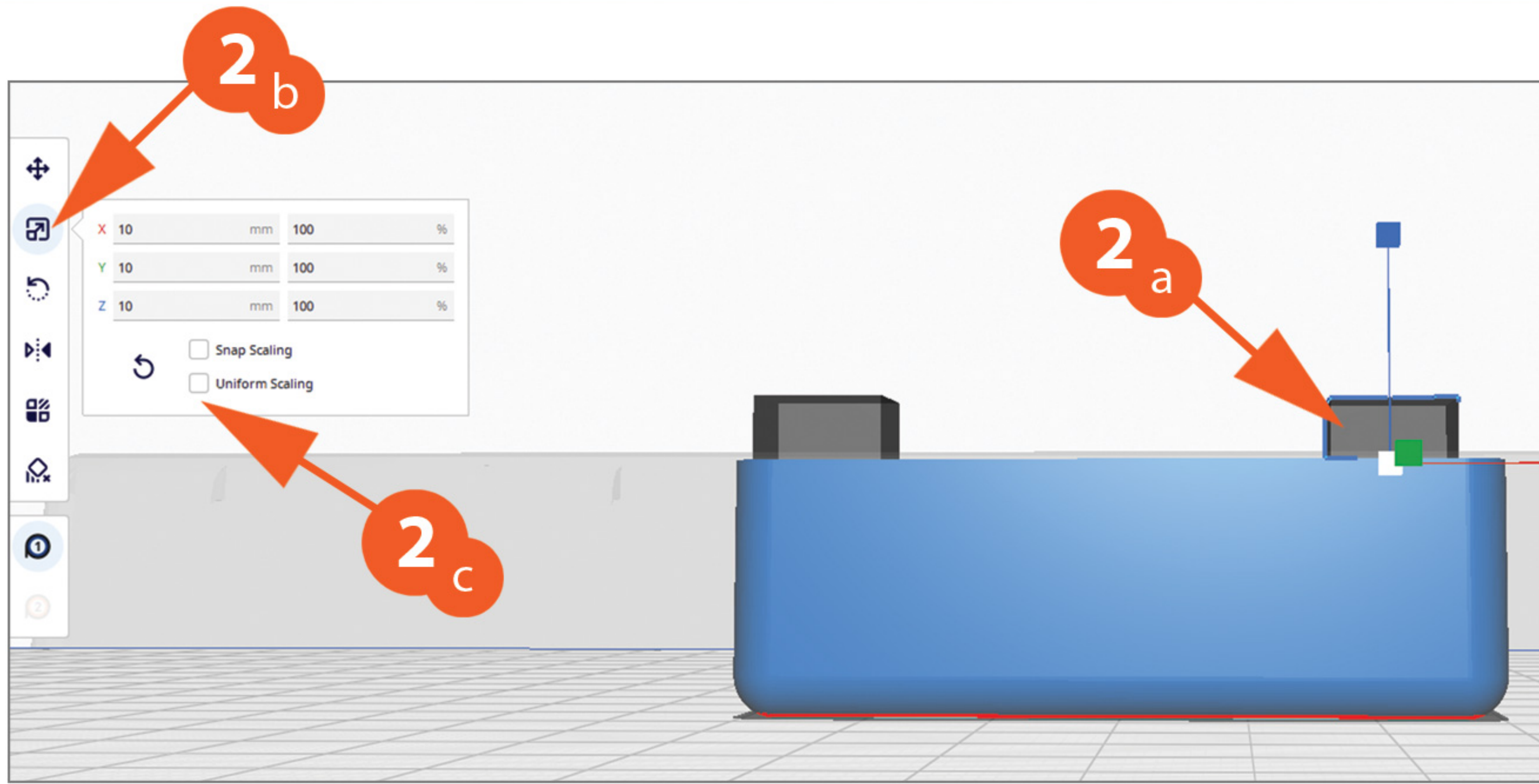
1d - Repeat for the second hole.



1e - Now from a top view, select one of the blocks. Move it so it is centered over the hole.

1f - Repeat with the second block.



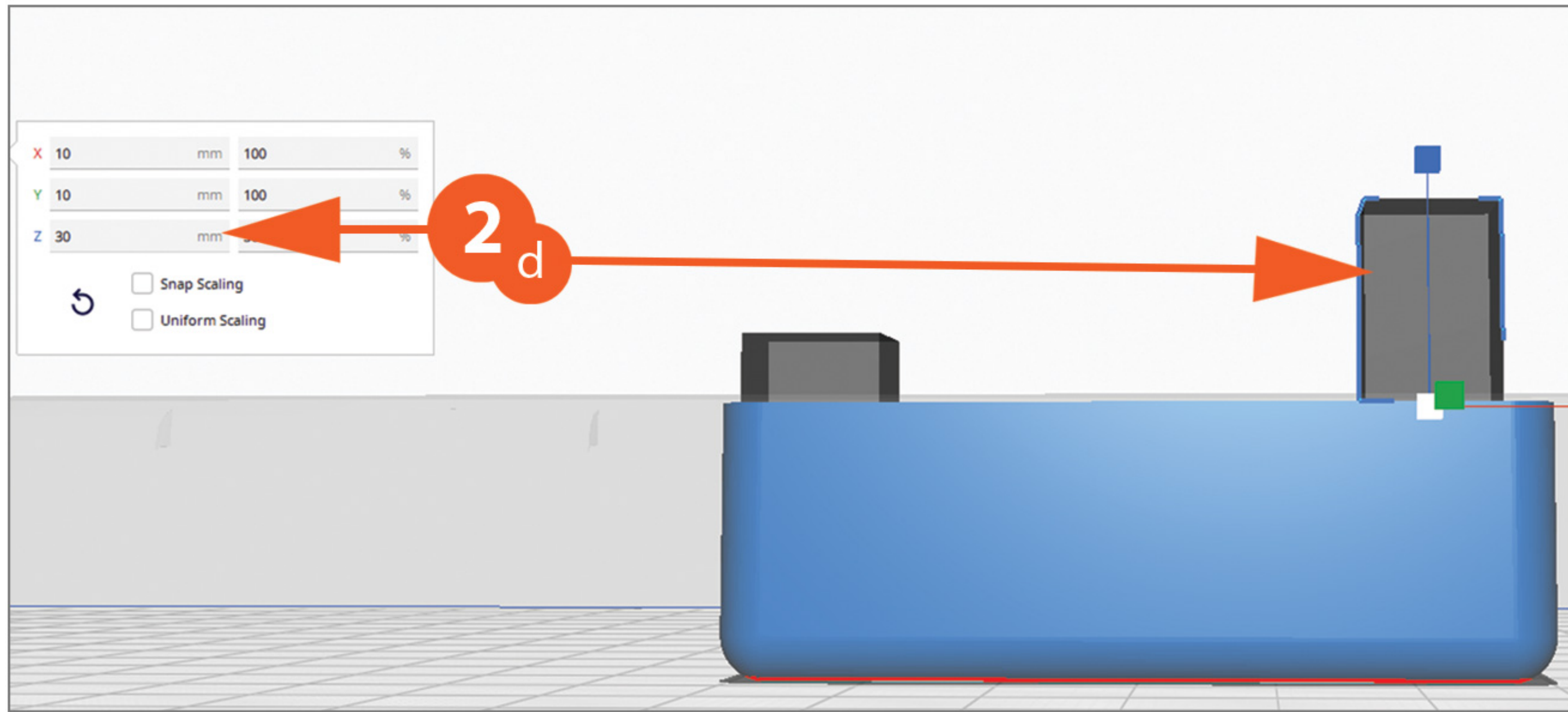


2

2a - Select one of the blocks.

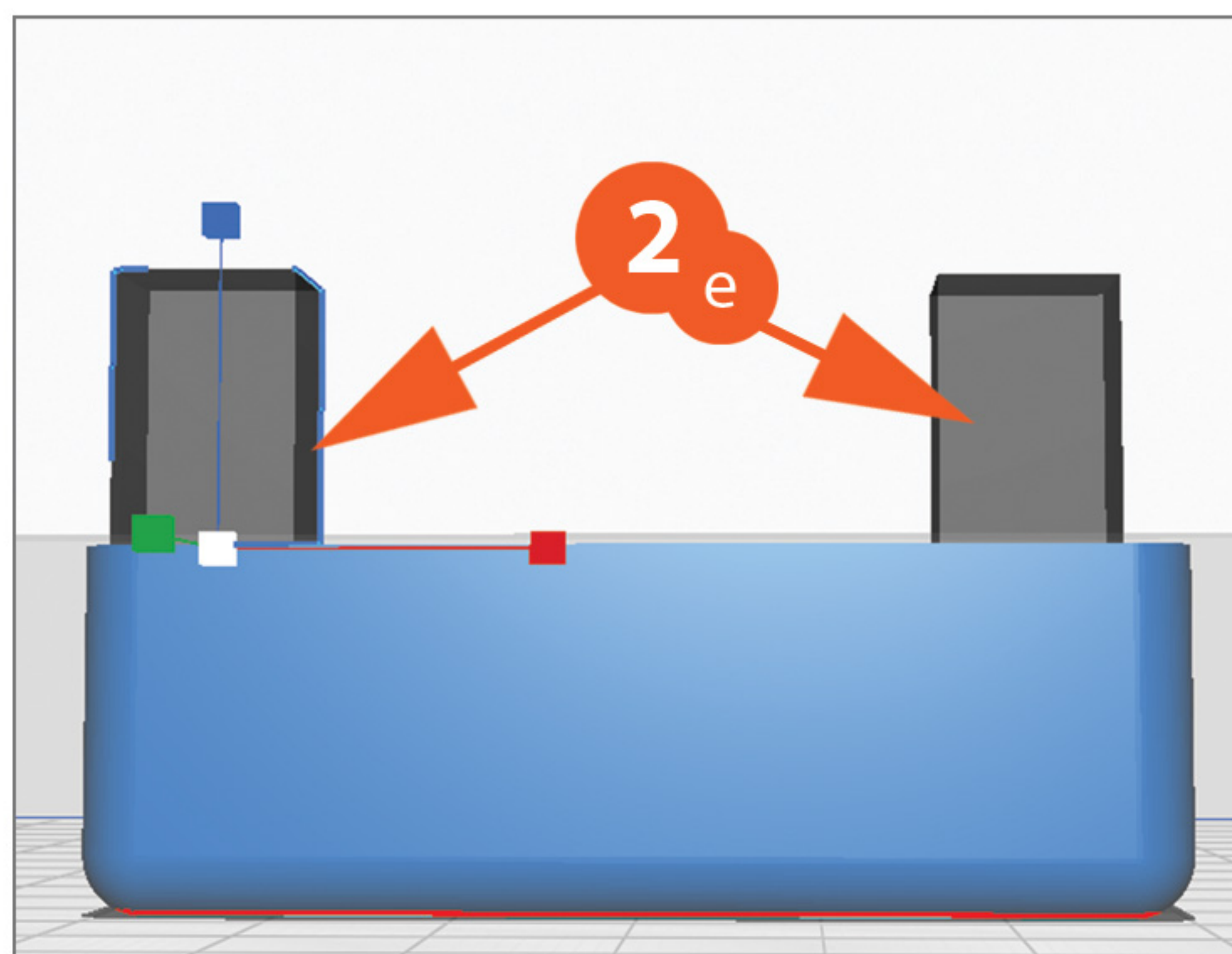
2b - Click on the **Scaling** icon.

2c - Uncheck Uniform Scaling.

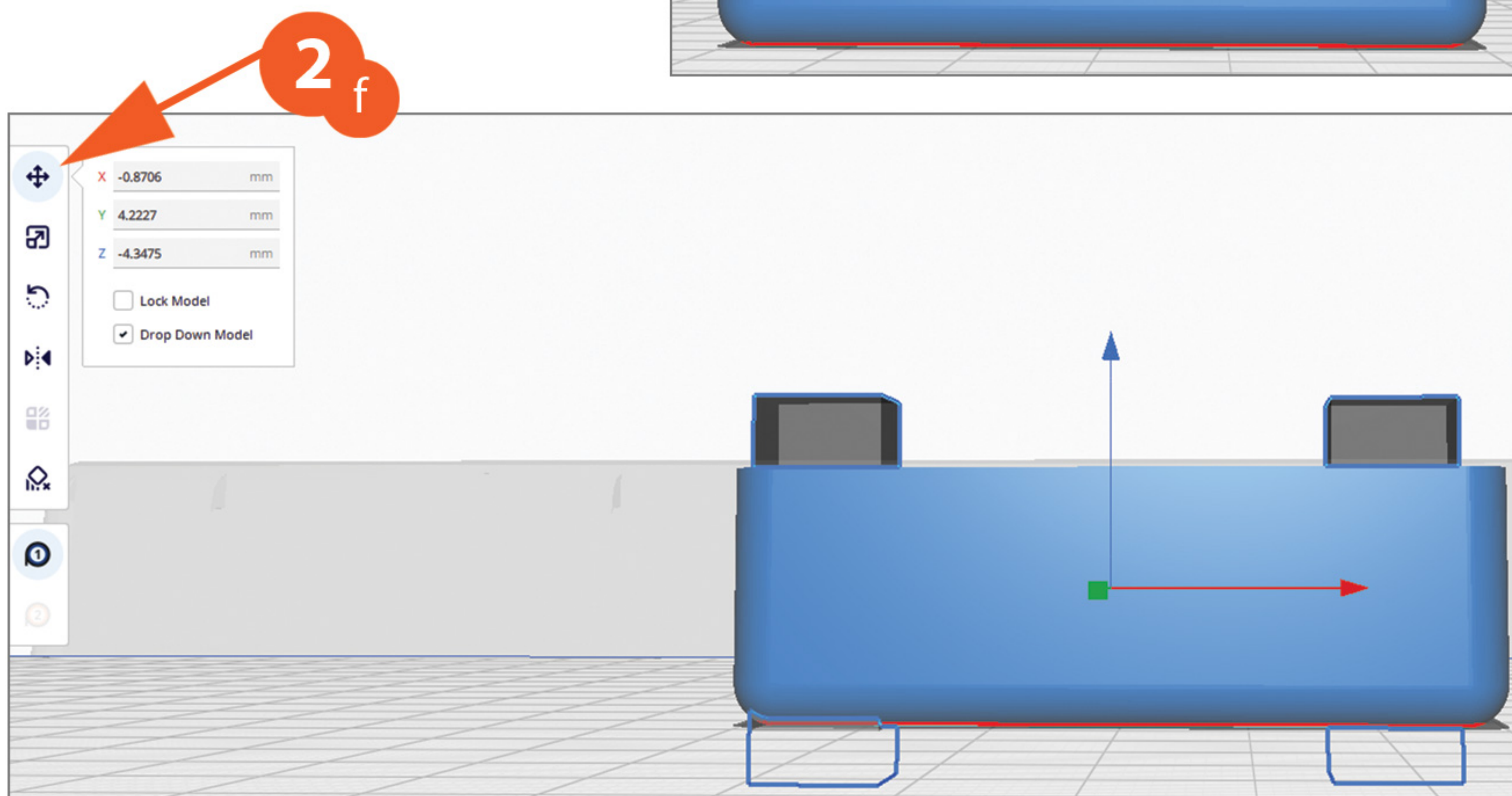


2d - Change the Z scale to 30 (or any number so the block is higher in total height than your object is).

Repeat this with the second block.



2e - Now select both blocks (hold shift while selecting).



2f - Select the **Move** icon, and move the the blocks down so they stick out of the bottom of your object.

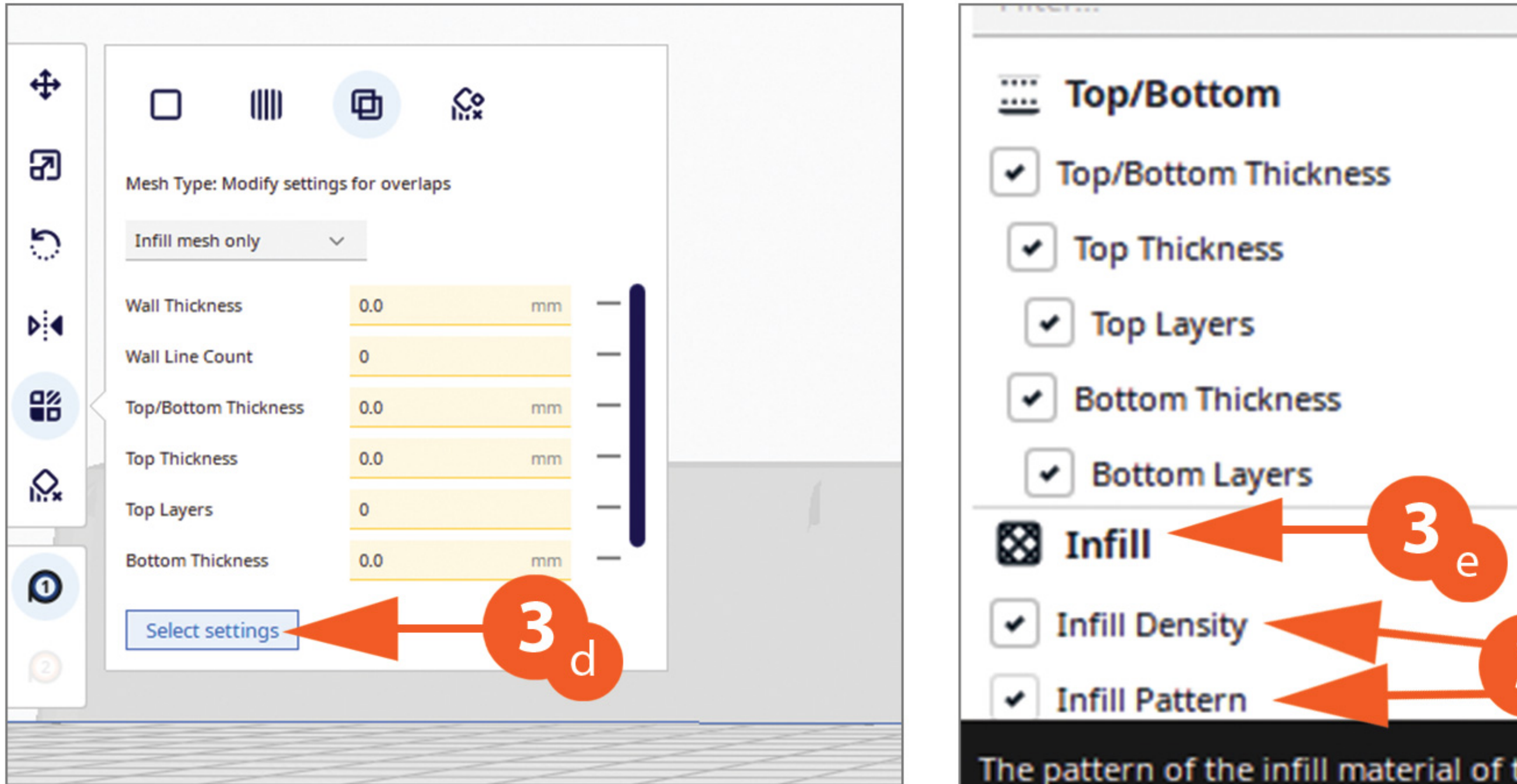


3

3a - Select one of the Blocks

3b - Select **Mesh** icon

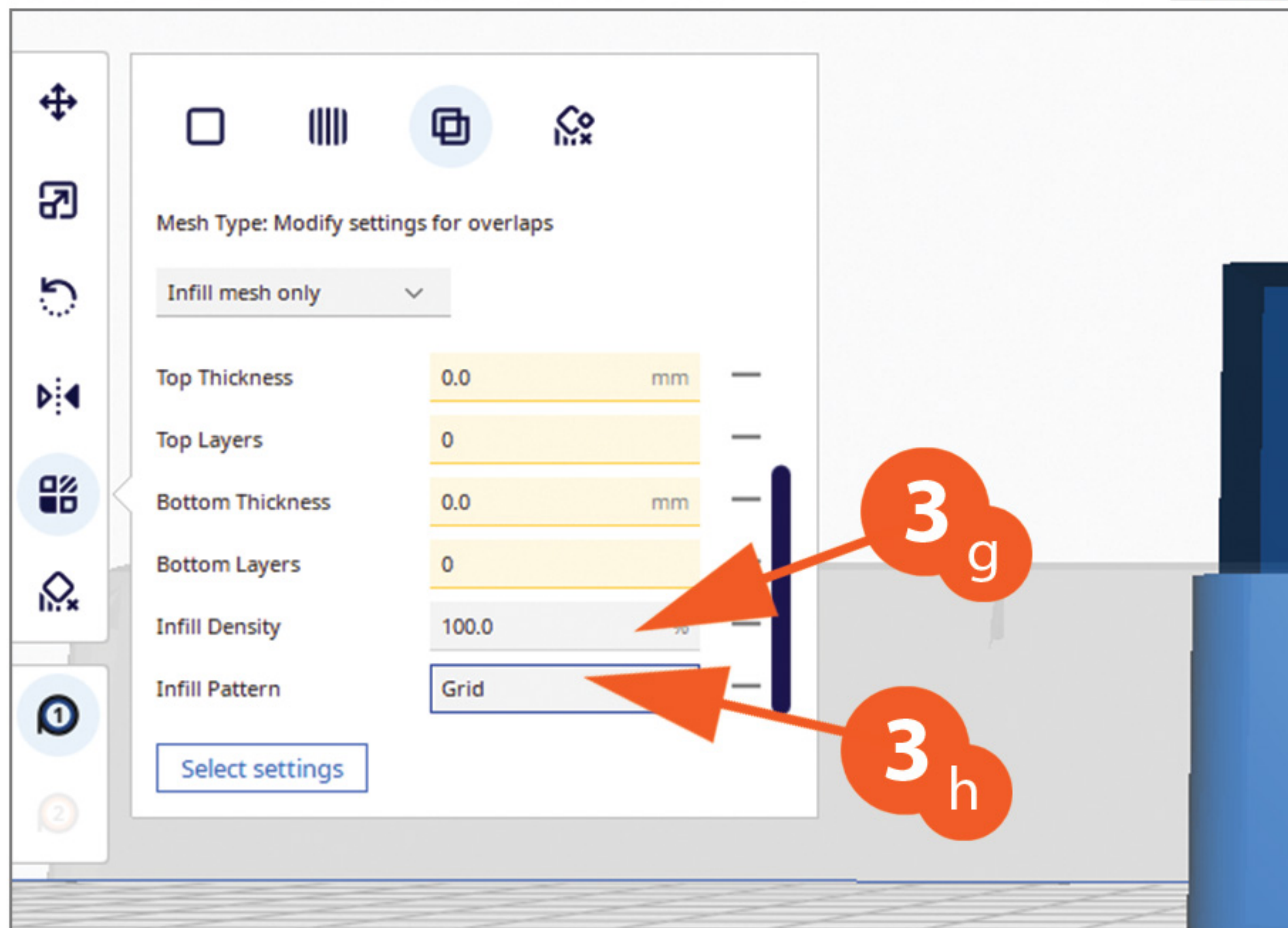
3c - Select **Overlap**



3d - On the Popup menu, Click on **Select Settings**

3e - From the second Popup menu, scroll down until you see **Infill**.

3f - Check **Infill Density & Infill Pattern**
Close this menu.

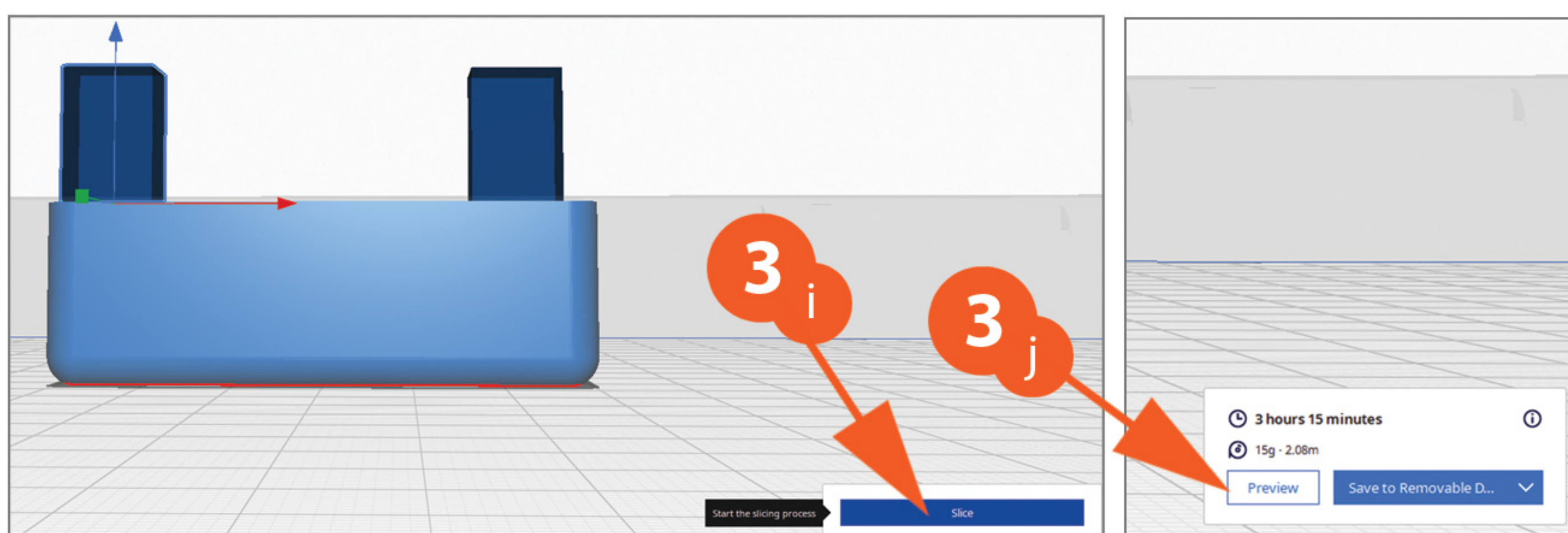


Now when you scroll down on the first Popup menu you will see Infill Density and Infill Pattern.

3g - Change Infill Density to 100%

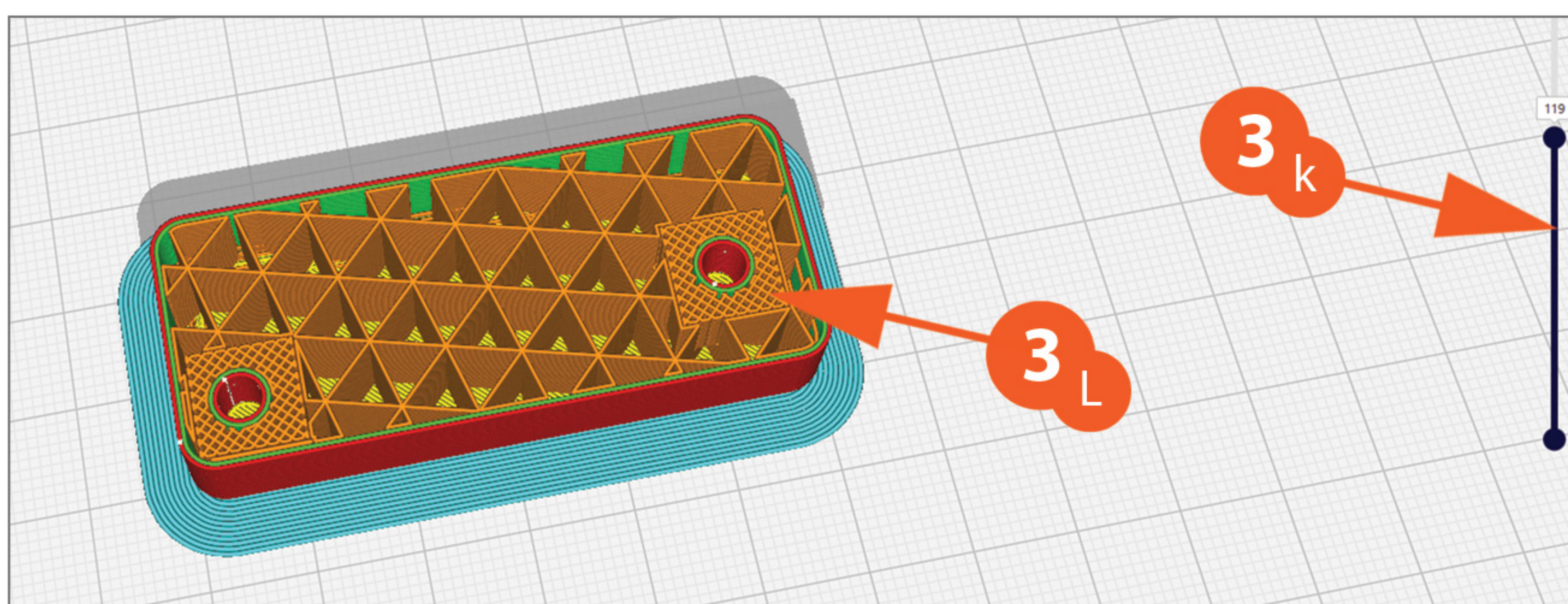
3h - Change Infill Pattern to Grid.

Repeat all of these steps for the second block.



3i - Slice your project.

3j - Click on Preview.



3k - Using the slider on the right, slide it down to see inside the mesh.

3L - You should see that the areas around the holes are denser than the surrounding areas.

Your project is ready to print.

HOW TO 3D PRINT WITH THE ULTIMAKER S5 USING THE CURA SOFTWARE

Operating The S5



Version v1

Published December 29, 2022

Author Don Gillard

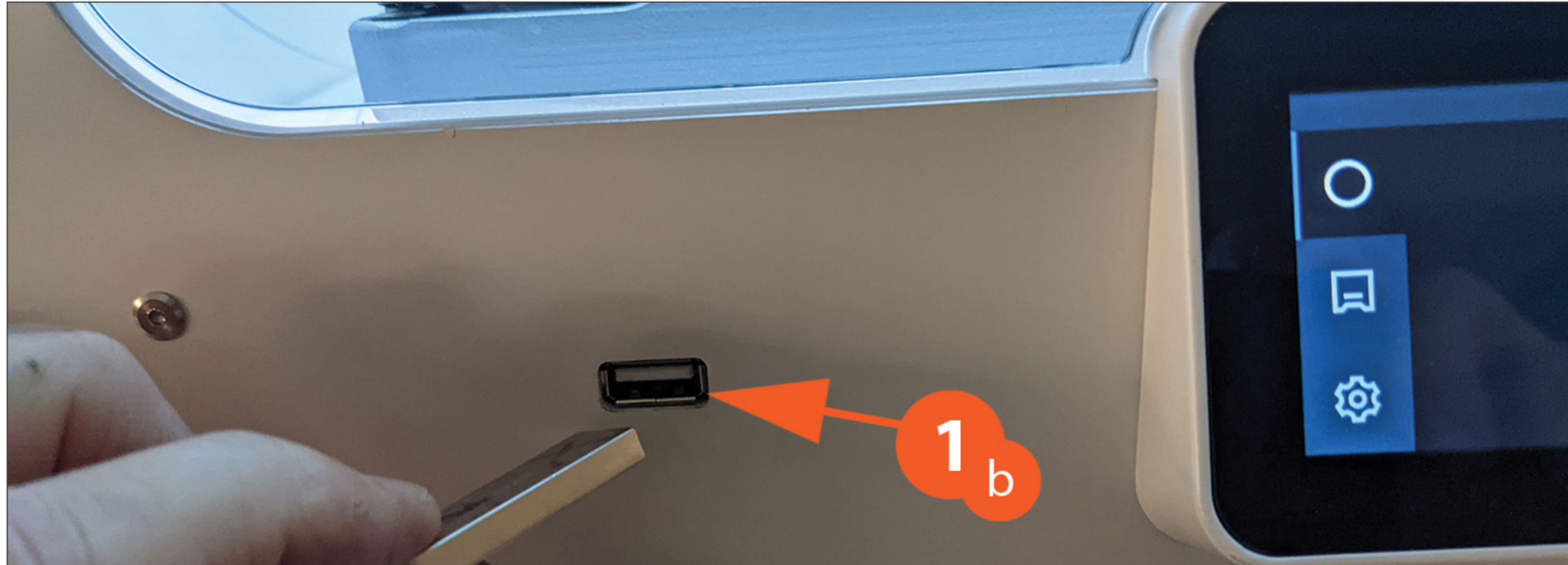
Langara College Makerspace

All rights reserved, Langara College 2022-23



1

1a - Turn on the Ultimaker S5, the power switch is at the back.
O is On, **I** is Off.

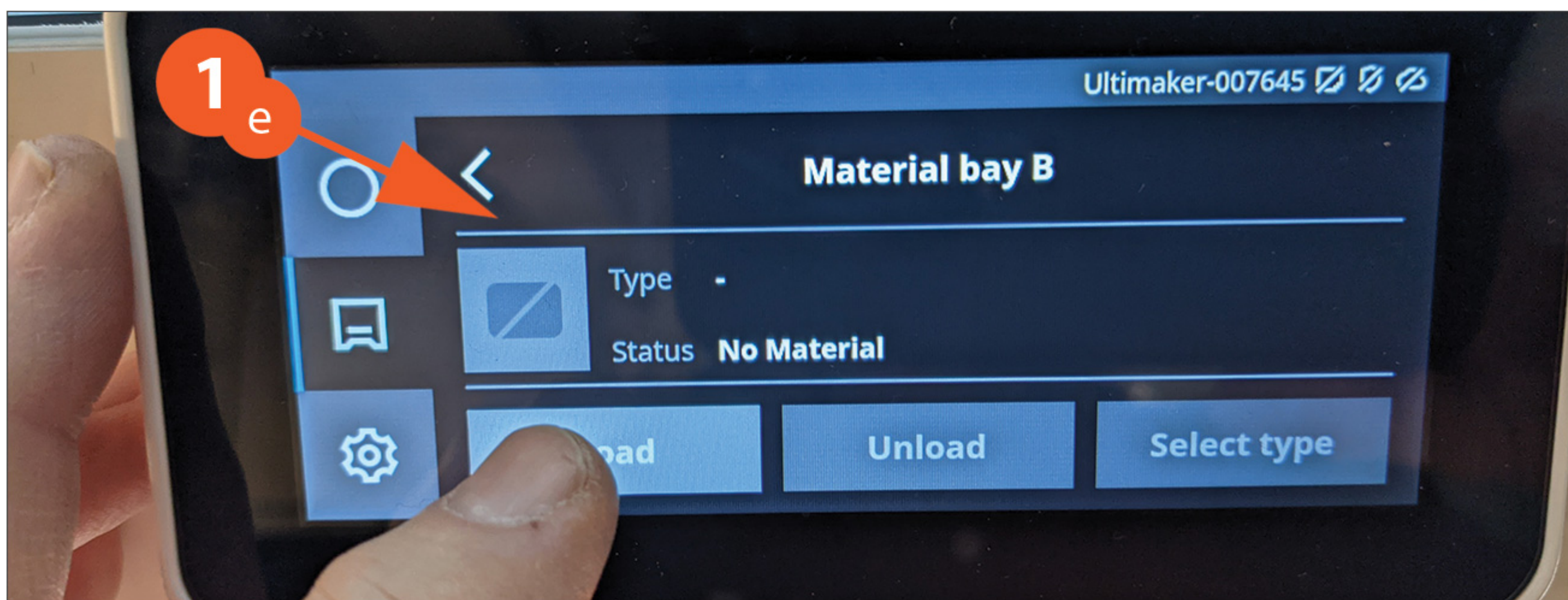


1b - Plug your USB into the front USB slot.



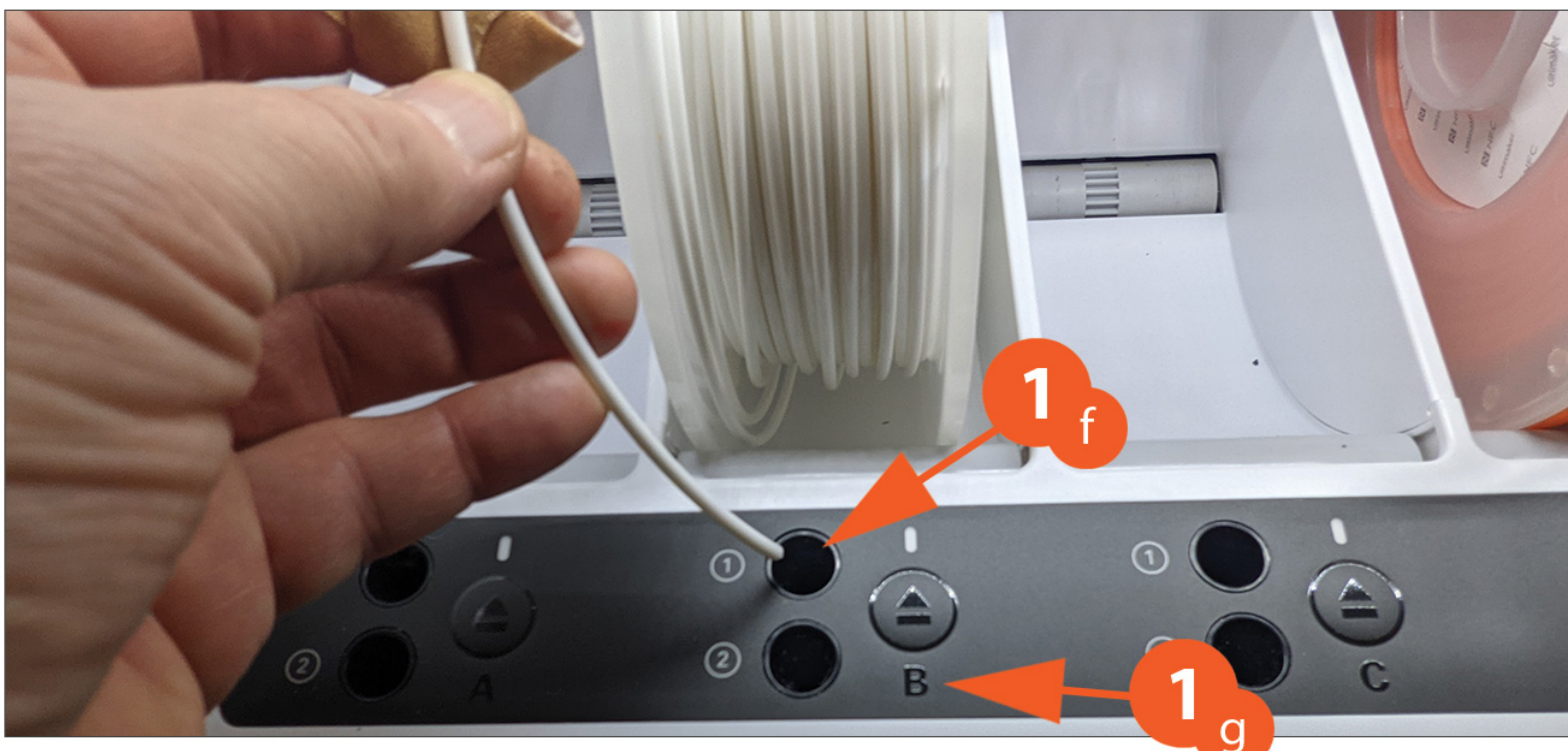
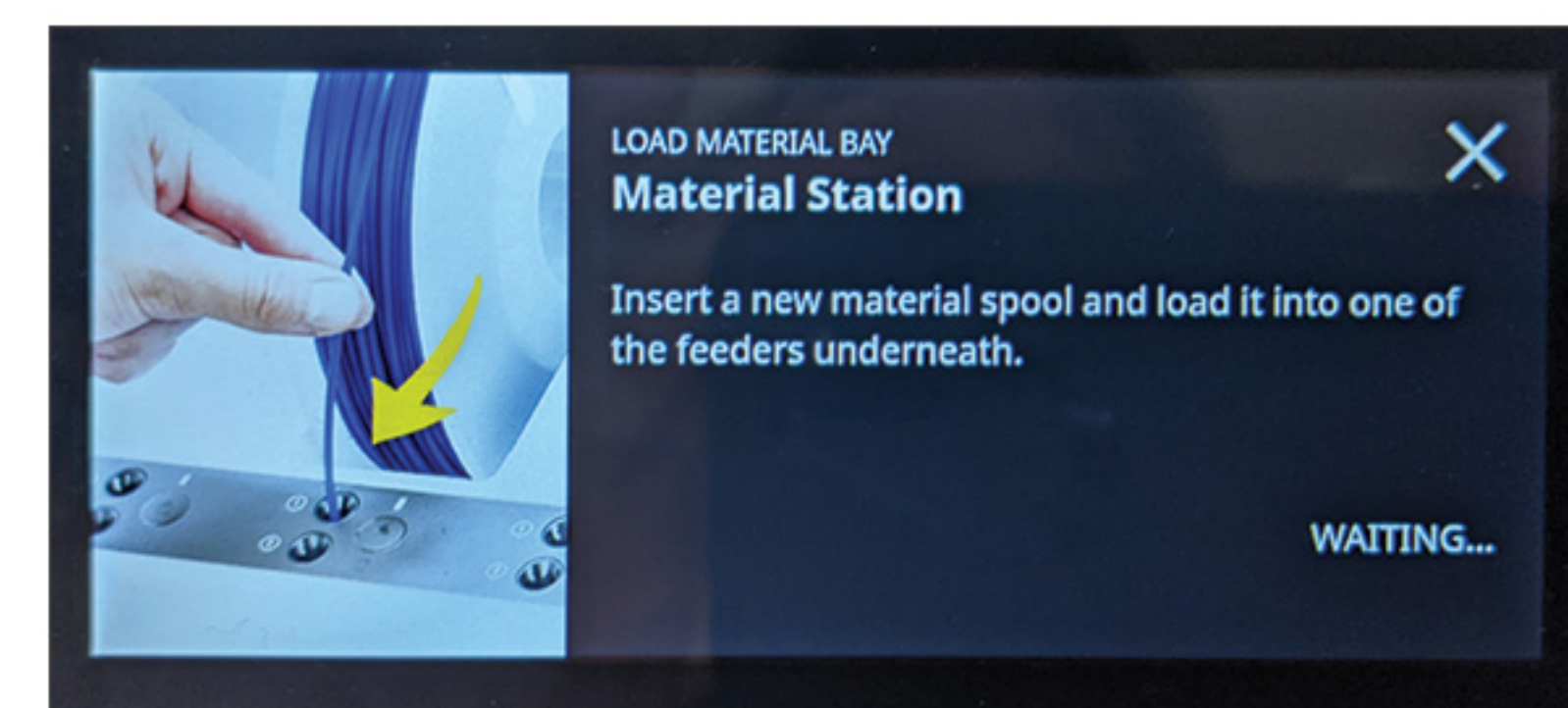
1c - Select the second **Icon** (printer icon).

1d - Then select the **Letter** where your first material is.



1e - Select **Load** material.

You will see this menu popup...



1f - Then open the Material Station door, and insert your material filament into the #1 hole...

1g - At the **B** material (or the letter that your material is in).

Then push it in until you feel it grab the filament.



2

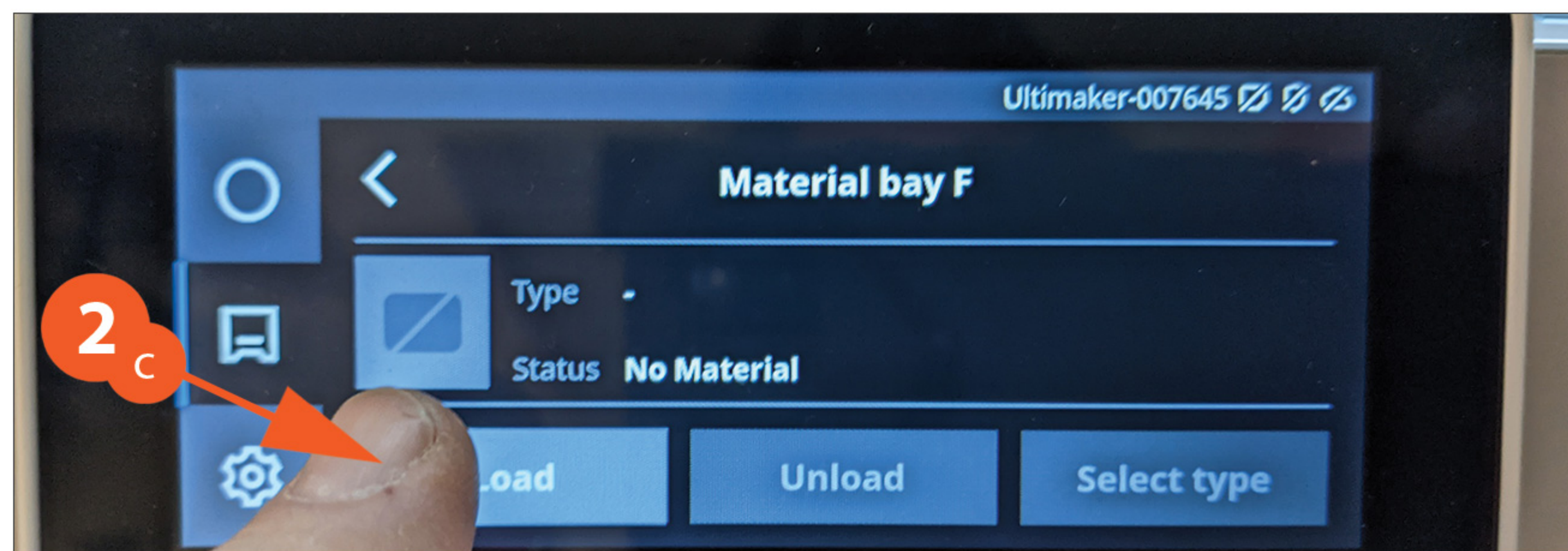
2a - You should now see head #1 - AA 0.4 linked to ABS White (or the colour you choose).

Please note:

You must use the exact settings on the S5 printer as your setup in Cura, otherwise it will not print.

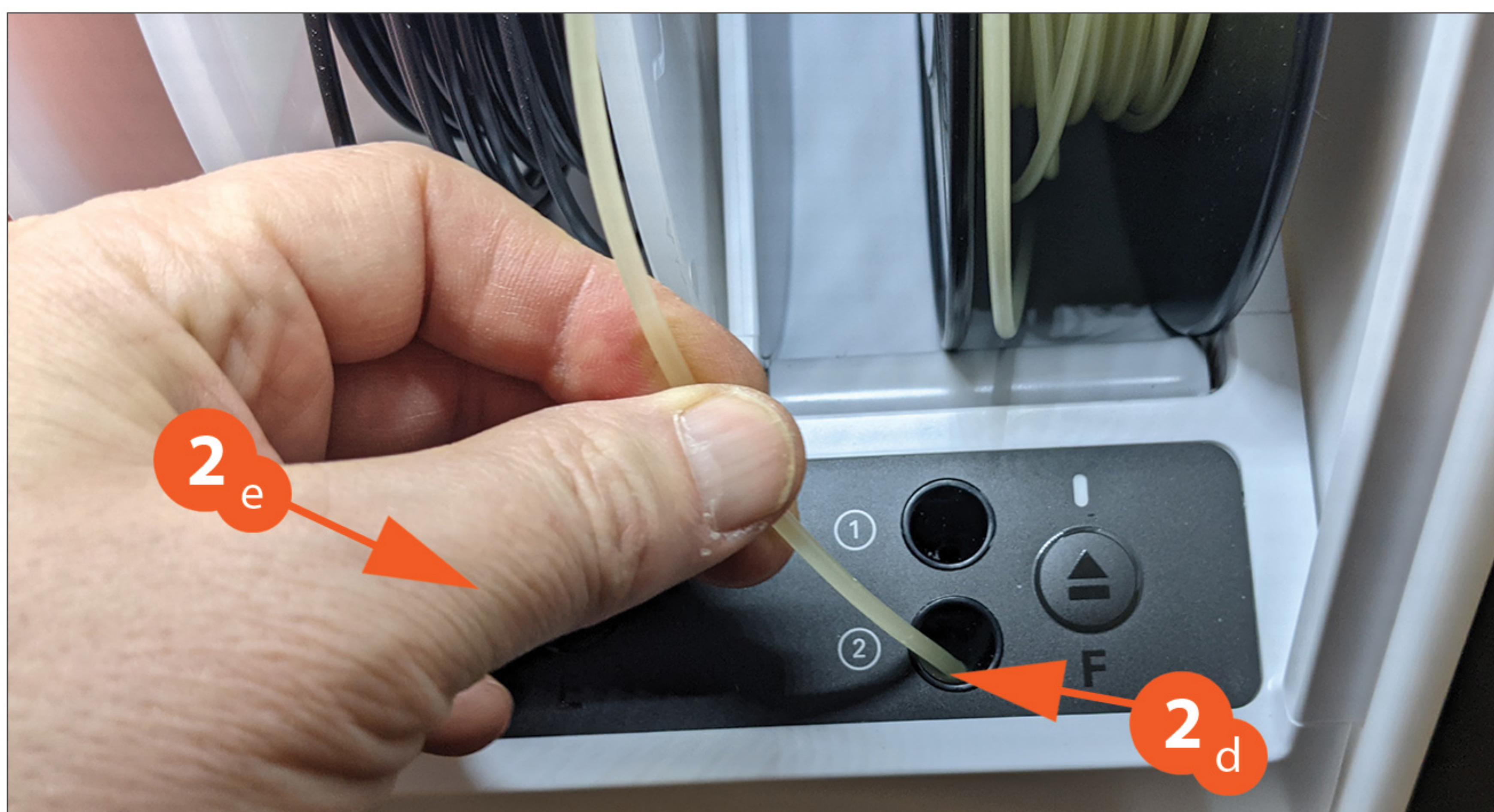
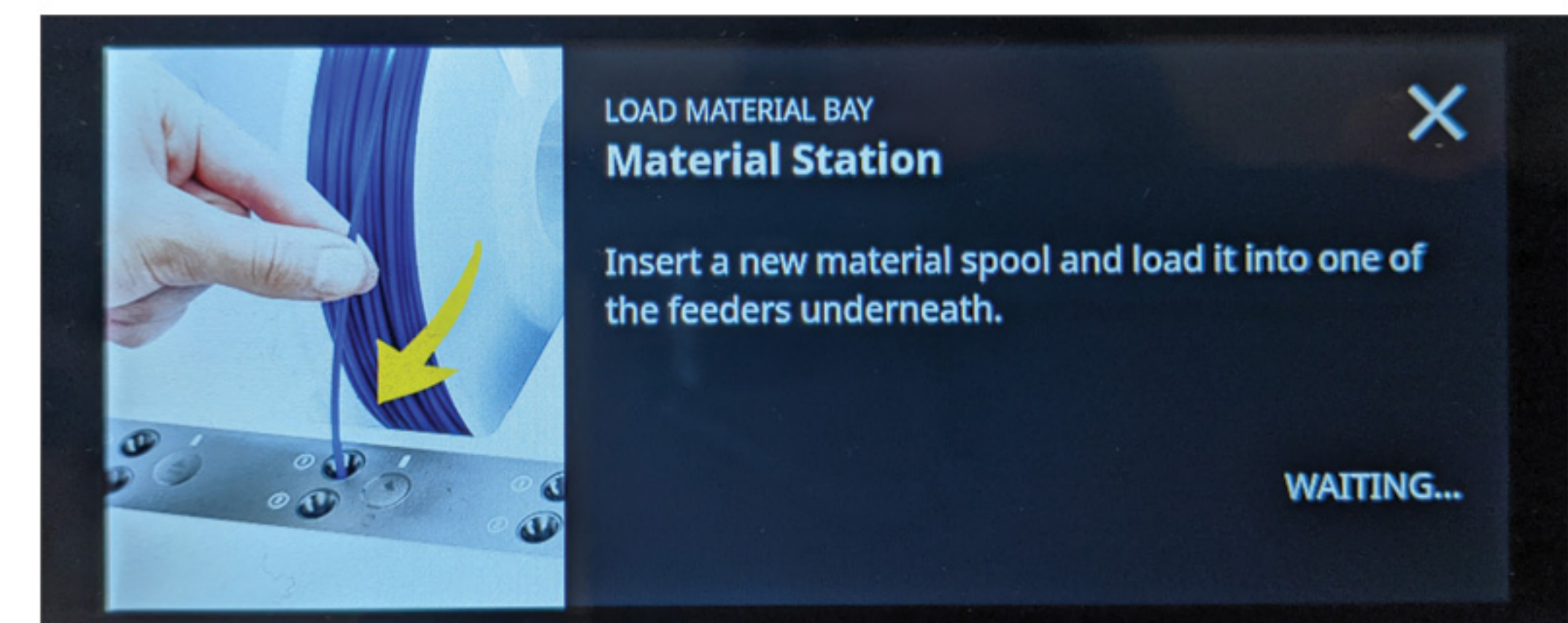


2b - Now select the **letter** where your second material is, or as in our demo the PVA.



2c - Select **Load** material.

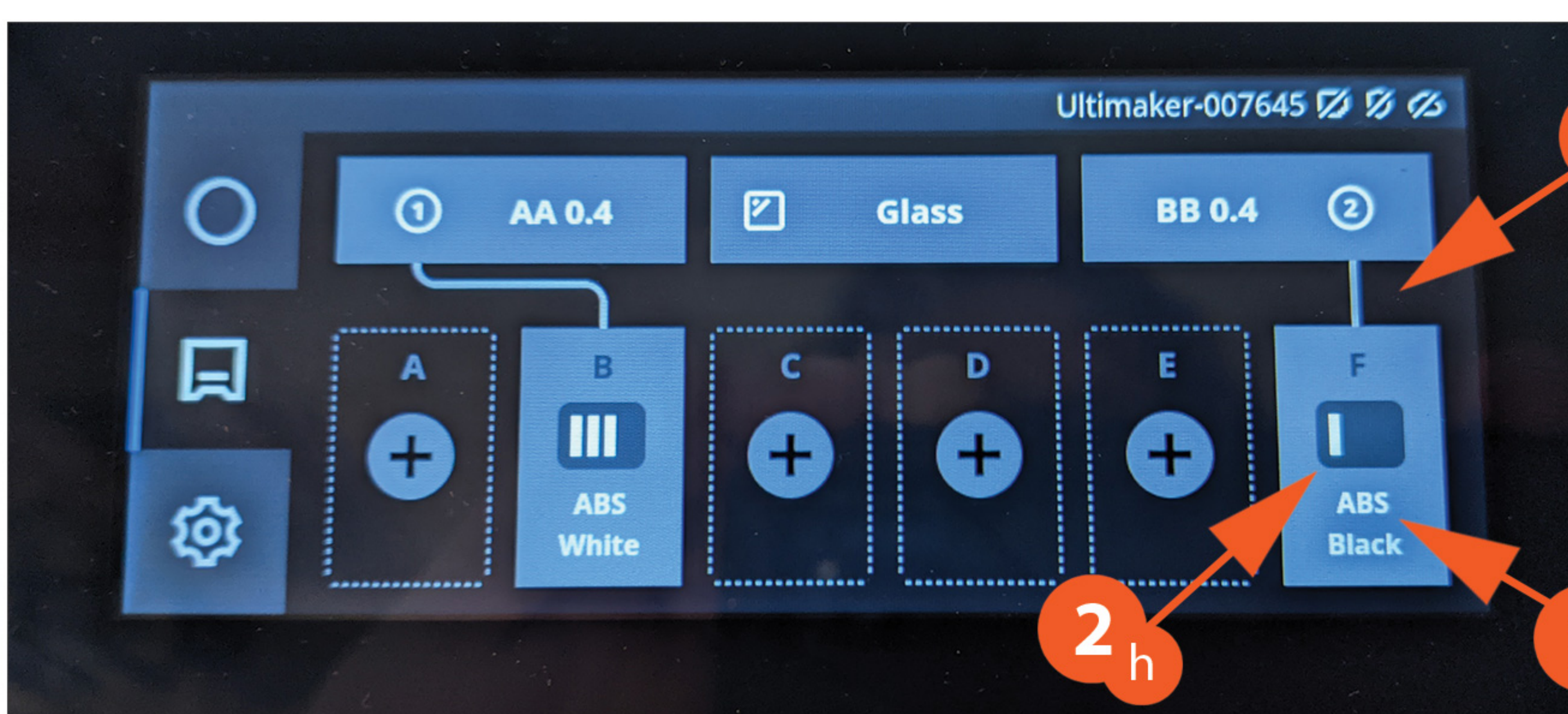
You will see this menu popup...



2d - Then open the Material Station door, and insert your material filament into the #2 hole...

2e - At the F material (or the letter that your material is in).

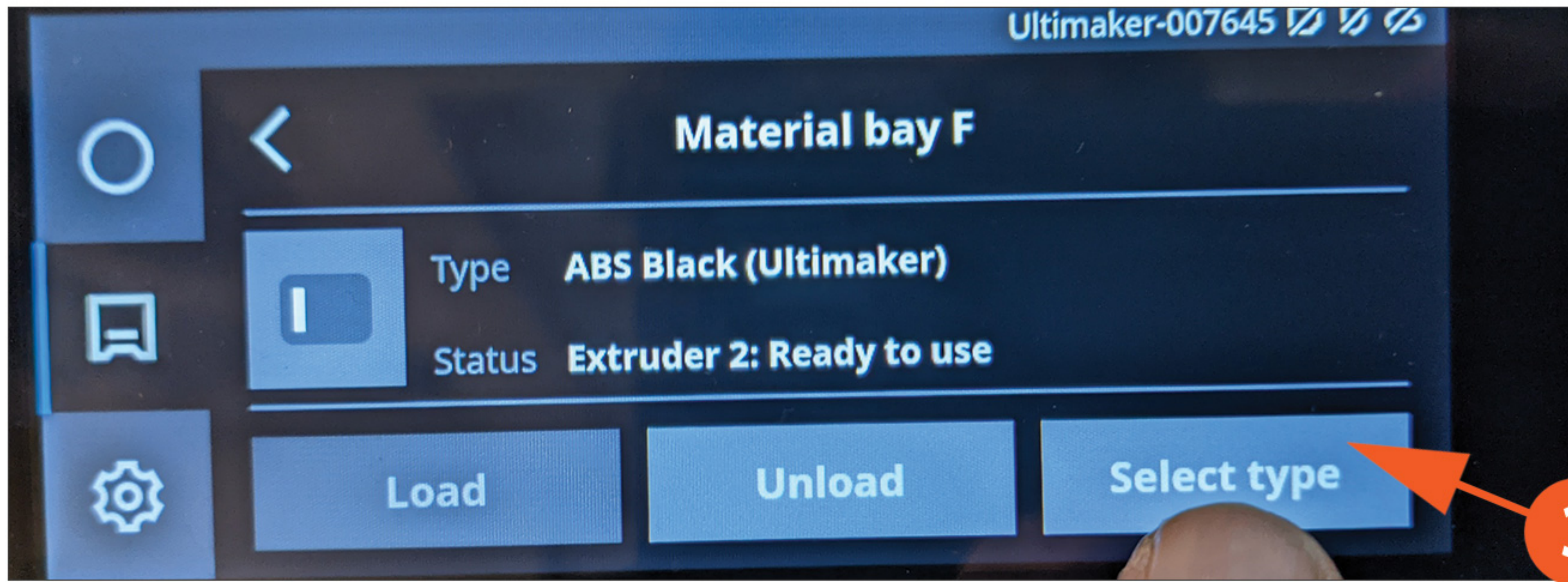
Then push it in until you feel it grab the filament.



2f - You will see this screen now...

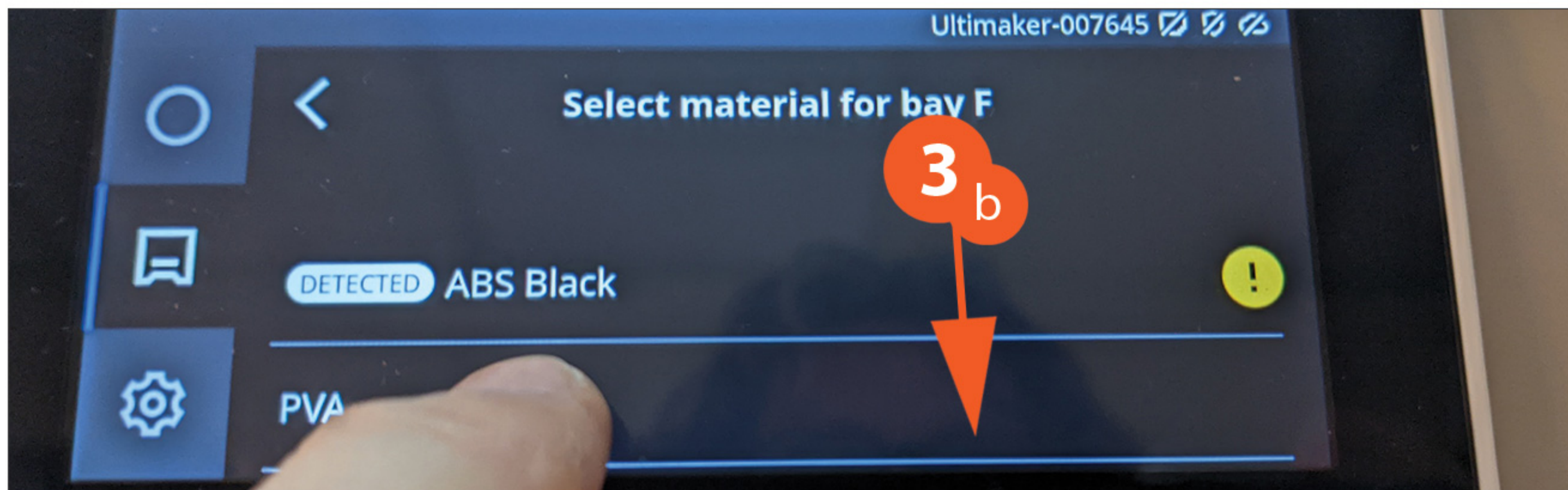
2g - And it may not be the material you loaded.

2h - To change the material, Click on the material box.

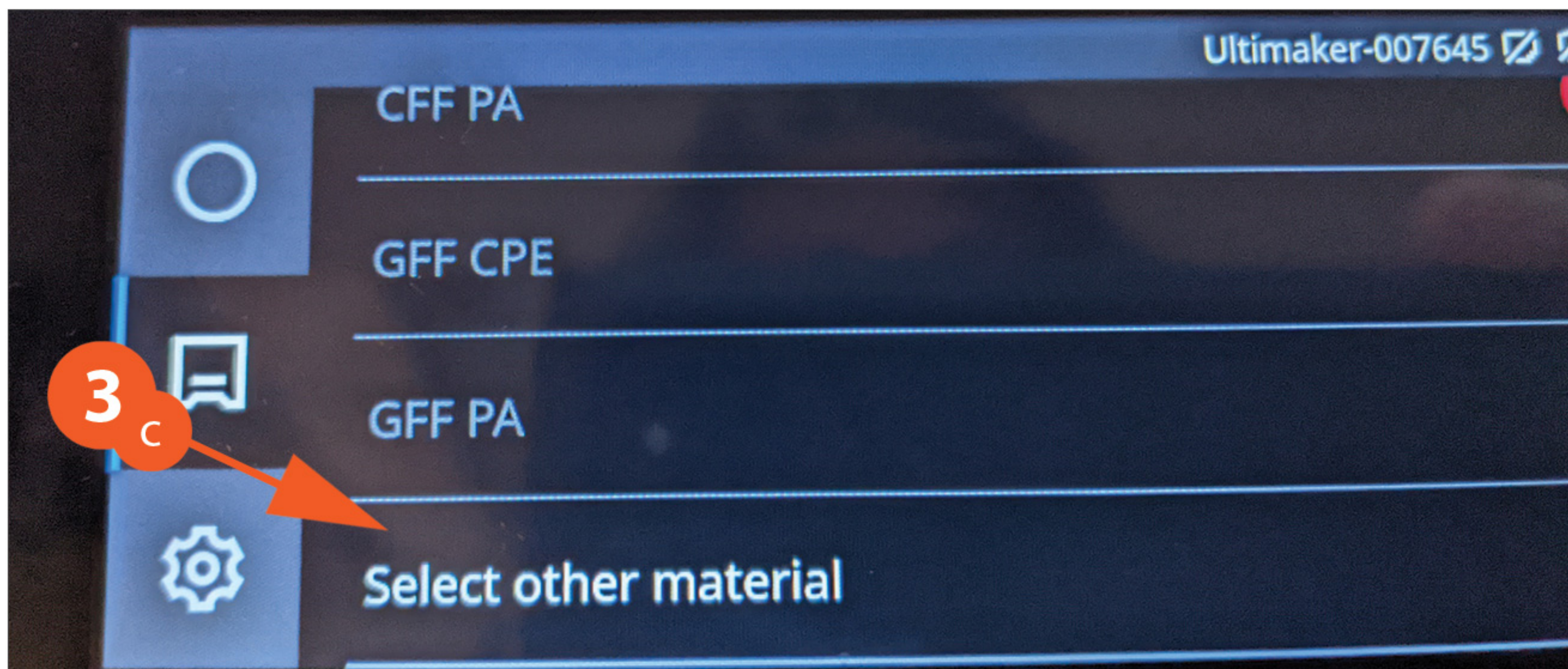


3

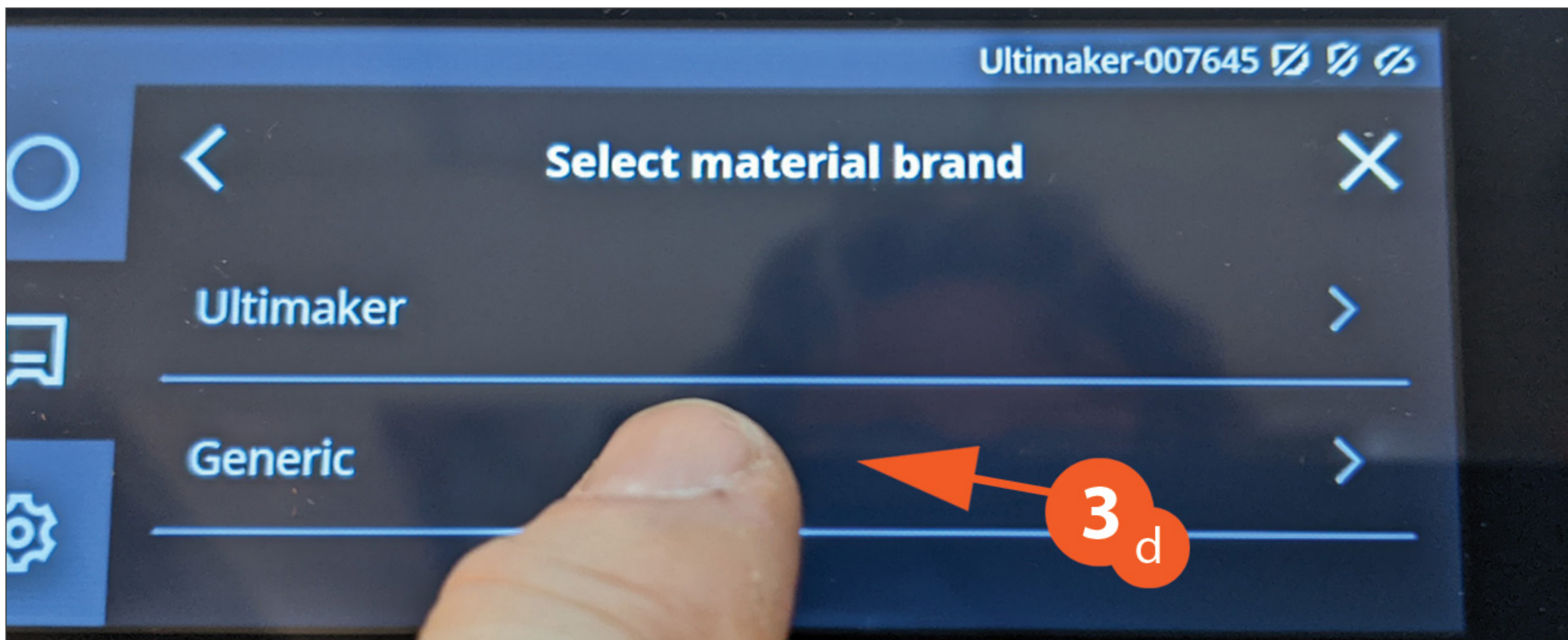
3a - Click on **Select Type**.



3b - Scroll down the list.



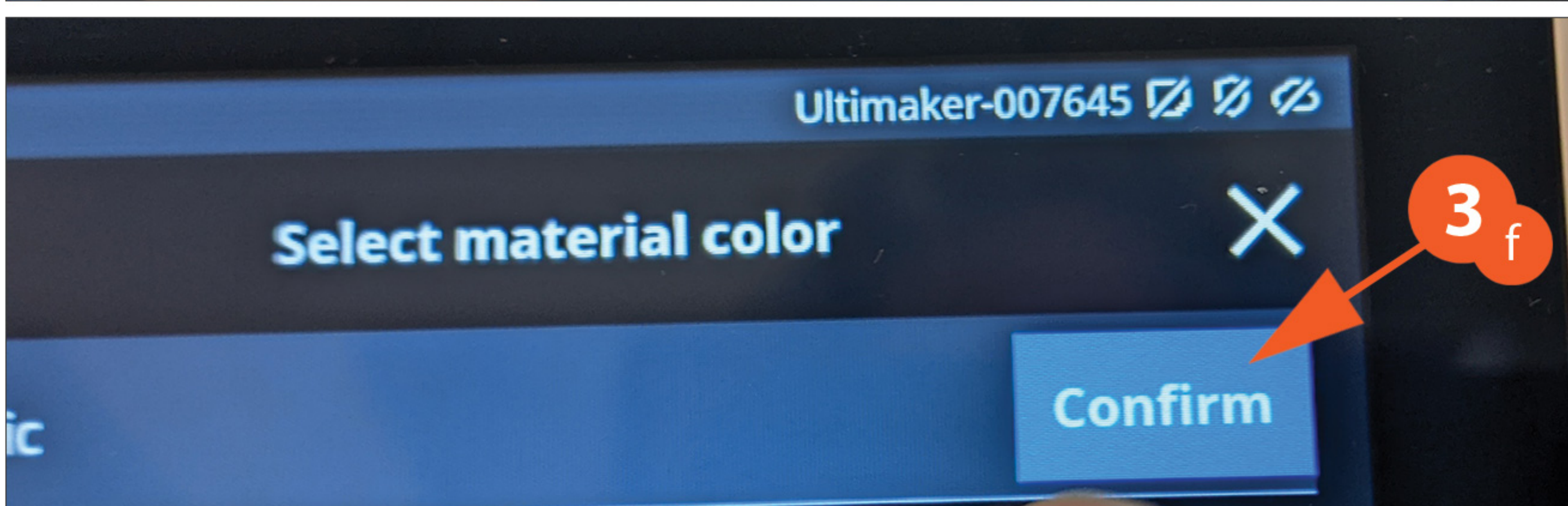
3c - To **Select Other Material**, click on it.



3d - Click on **Generic**.



3e - Then click on **PVA**.



3f - Click on **Confirm**.



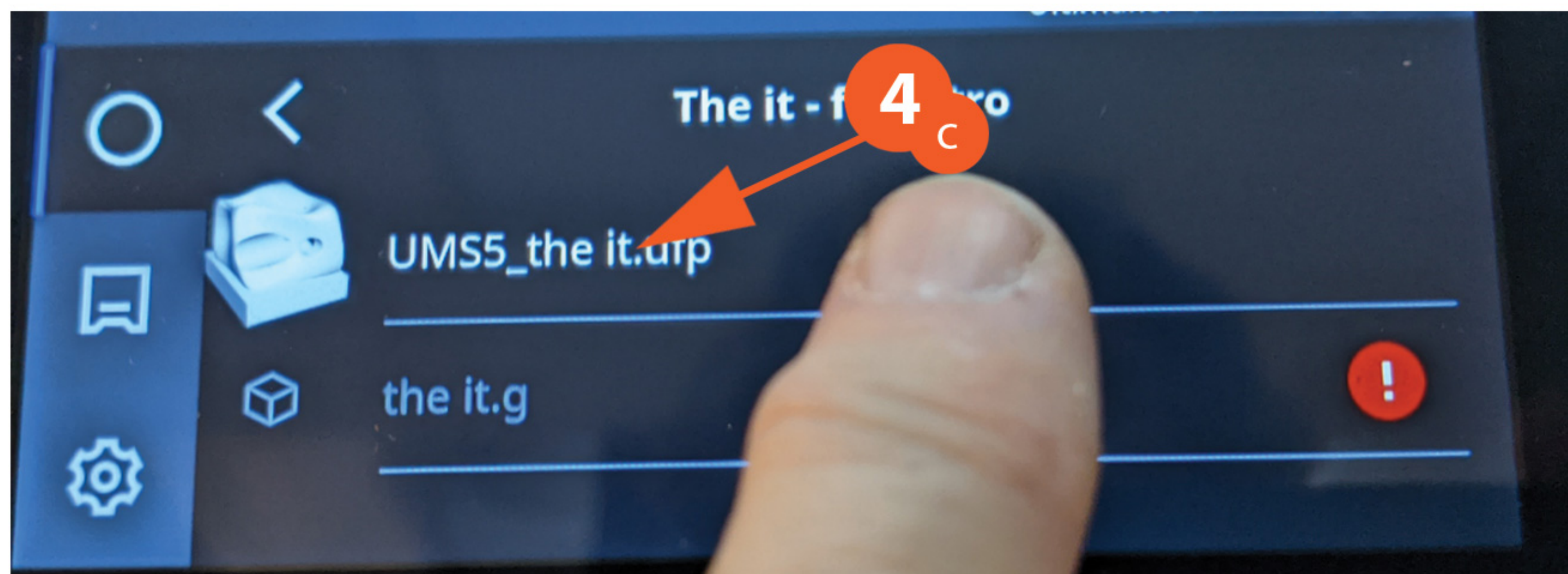
4

4 - You should now see both of your materials on the **Print Screen**.

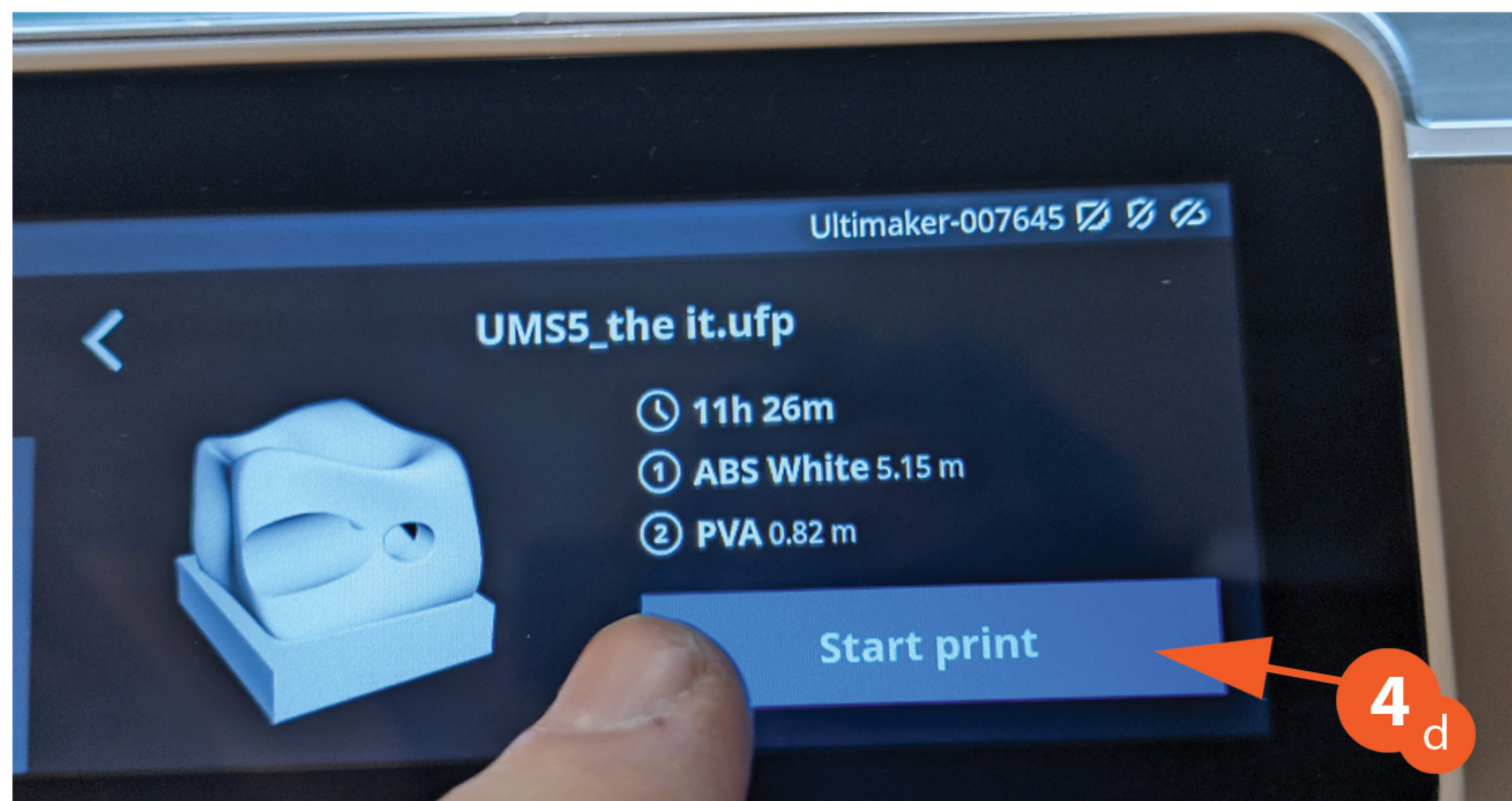


4a - To start your print, Click on the first top left icon.

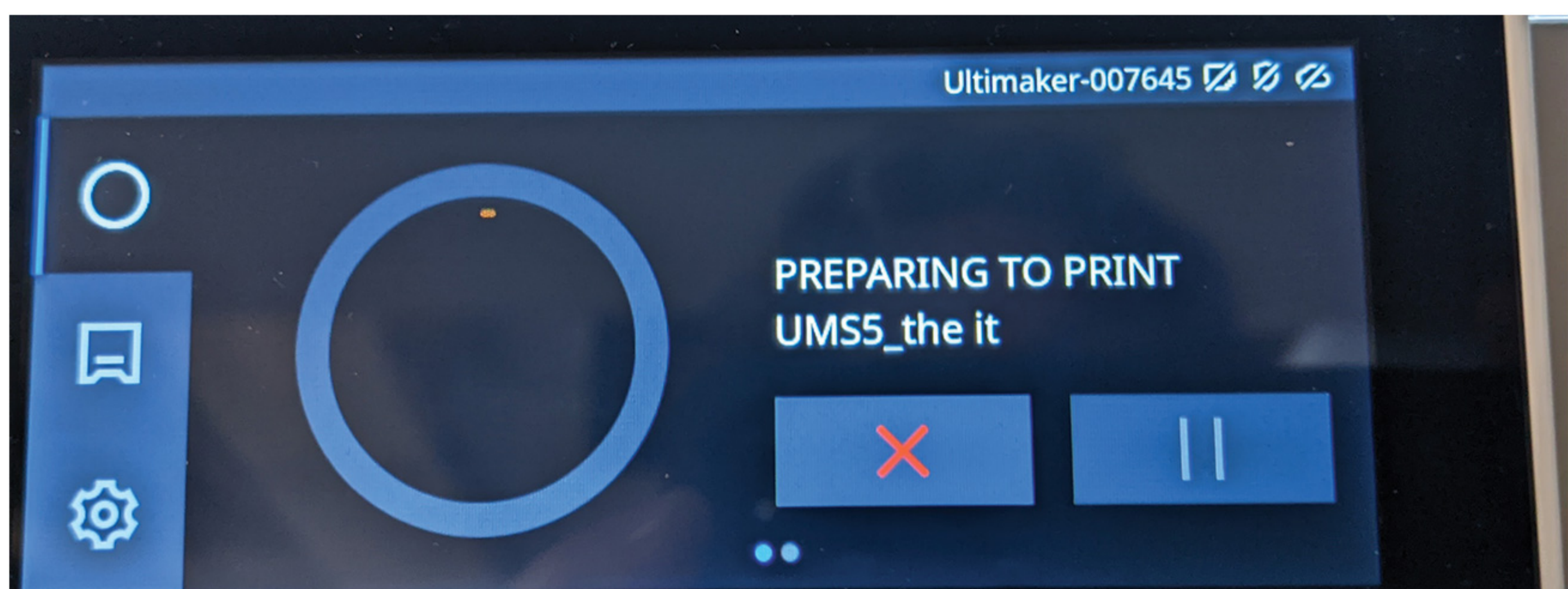
4b - Click on the **Select From USB** button.



4c - Click on your file.



4d - Click **Start Print**.



If you do not have the same materials loaded as specified in your print, the print will not start.

Keep an eye on your print for the first hour. If it is peeling up, warping or not sticking to the glass platform you will need to stop the print.

Ask the tech how to proceed if this happens.