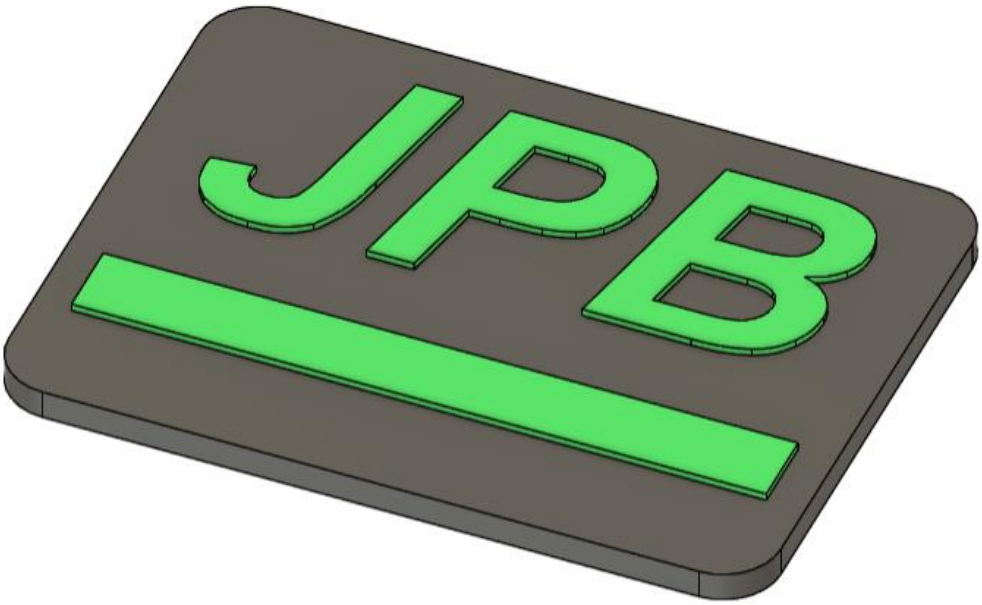
Mr. Barbetta's *"I don't care about 3D Printing. I just want the art credit."* class

# Make a Dual Color Print



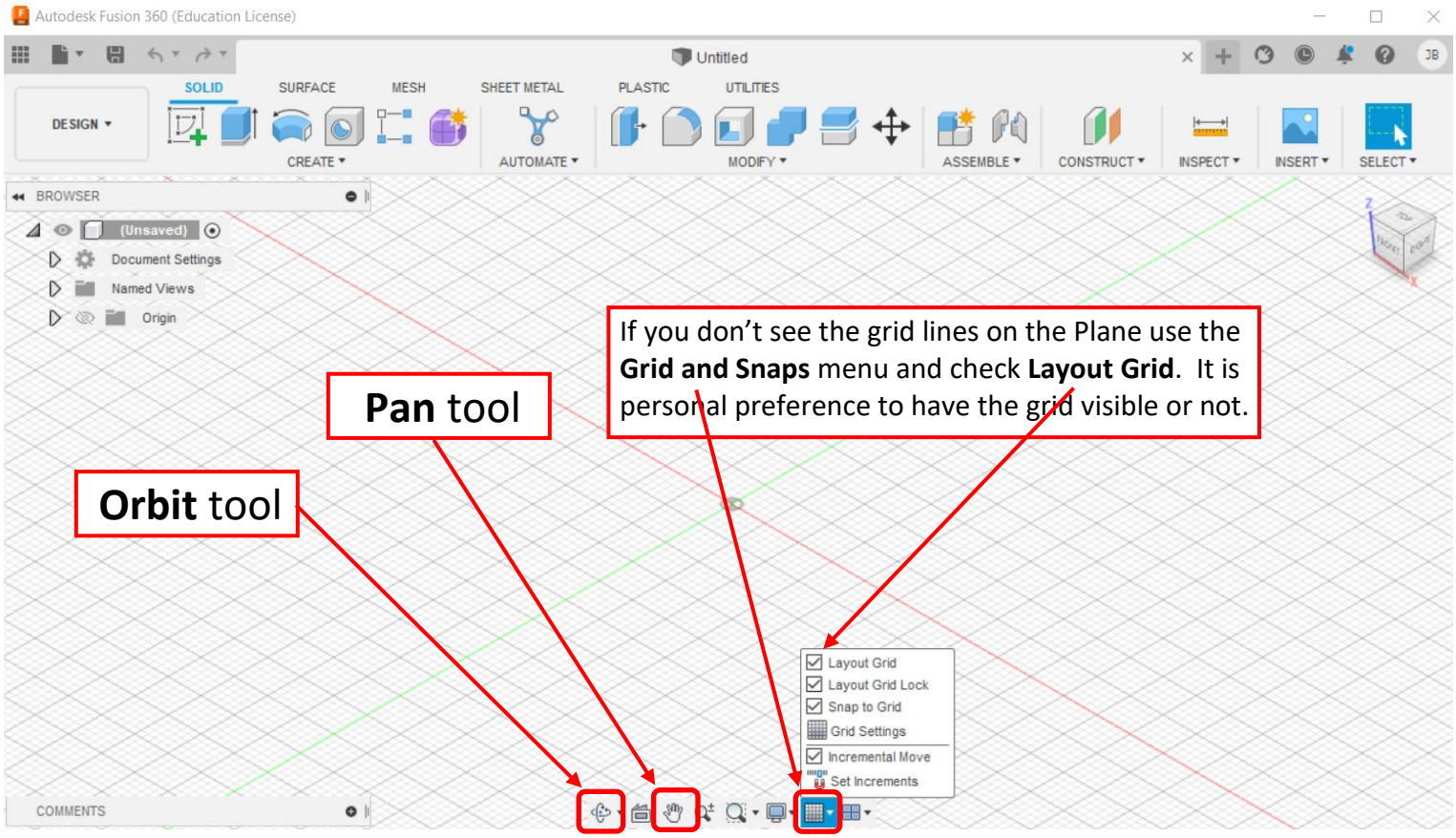
# Today's Lesson is Sponsored by Harbor Freight

# Contents

# Changing the View of a Design

- if you don't see a grid in the Fusion 360 window, as shown below, click on **Grid and Snaps** and check **Layout Grid**. Displaying the *Layout Grid* is a matter of preference. When designing for 3D printing, it can be used to represent the *build plate*.

- click on the **Orbit** tool and click somewhere on the **Grid** to practice rotating and changing the angle of the view.
- click on the **Pan** tool and then on the **Grid** to practice moving the view laterally.
- after using the *Orbit* or *Pan* tool one must press the **Esc key** to exit that mode.
- use the **Mouse Wheel** to practice Zooming in and out.




Here is a close-up of the View Cube at the top right of the window.

- click on the **View Cube** and move the cube while holding the mouse button down. This is another way to rotate the view.
- click on the Top of the View Cube and note how the view just jumped to a Top View.

The View Cube now resembles that on the right.

- click on the **Curved Arrows** at the upper right of the View Cube and practice Rotating the View.
- click on the **Arrows** at the sides of the View Cube to practice jumping to various Views.
- click on the **Home** icon to the upper left of the View Cube. This can always be used to reset the view to the Home View

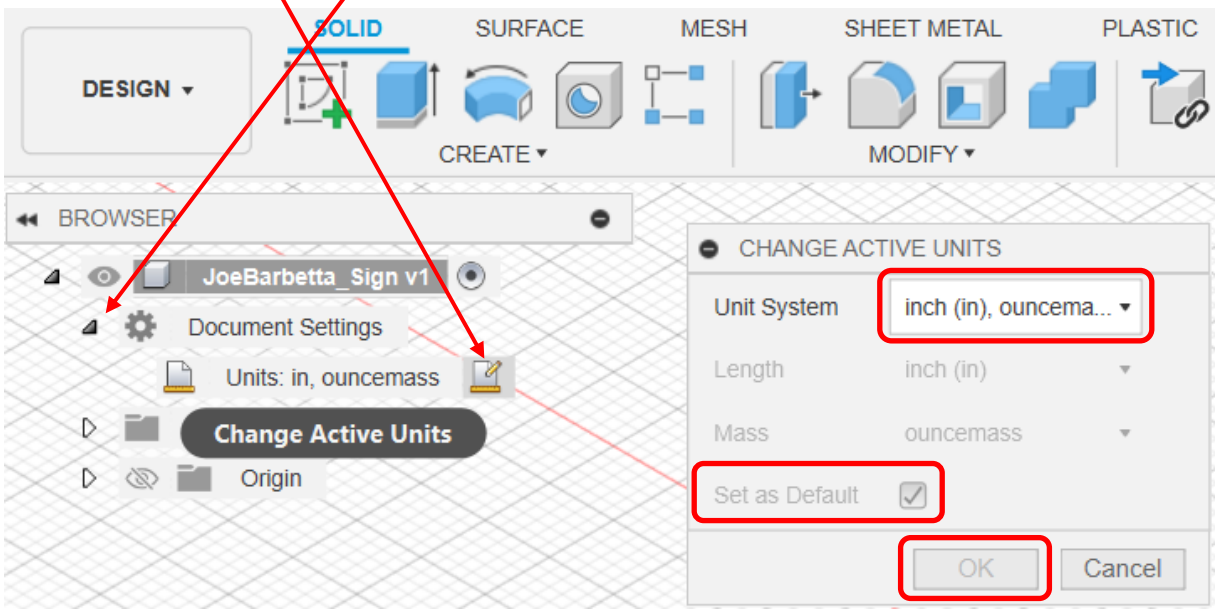
# Starting a Design in Fusion (START HERE)

- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_Sign** e.g. **JoeBarbetta_Sign** (note the use of the underscore)

Note that by default Fusion saves your project to "the cloud", which are the servers managed by AutoDesk. When you log into Fusion on a different computer, your projects will be available.

As you work you may want to occasionally save your work in case Fusion crashes or we lose power.

- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the right when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.



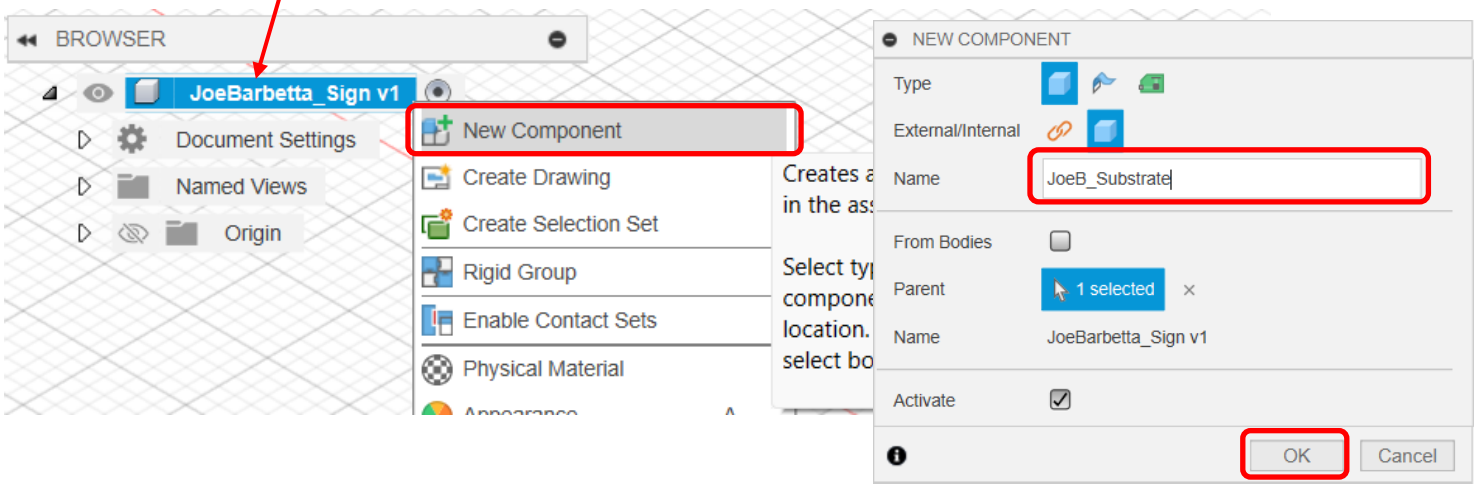

Note that the default units are in mm, which we just changed to inches.

Did you know that the default units have changed over the years? The earliest version used cubits as the default unit.

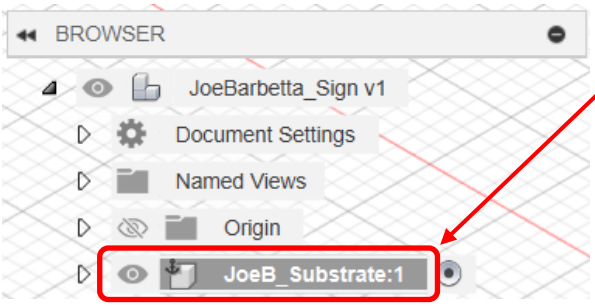
## Creating the Substrate Component

To keep a project organized it is recommended to create a new Component for each part.

- right-click on the **Project Name** at the top of the BROWSER and select **New Component**
- set **Name** as **Your first name and last name initial followed by _Substrate** e.g. **JoeB_ Substrate** and click **OK**

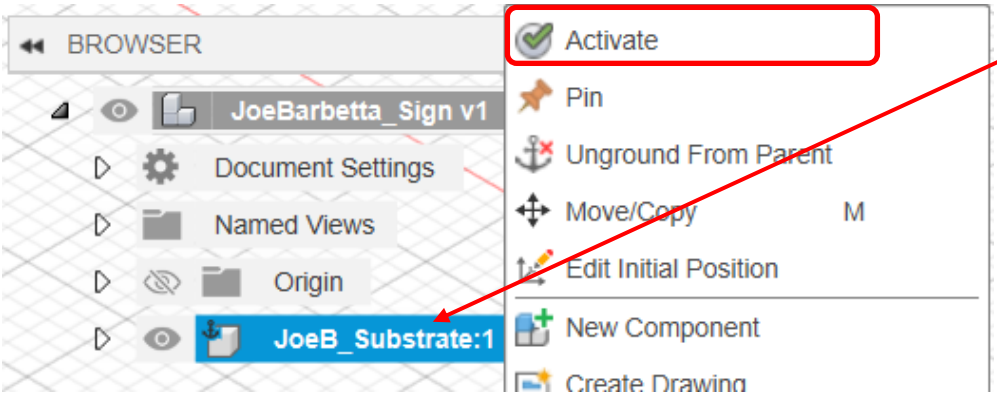



The new *Component* should show in the *Browser*.
Note that the area around the name is darkened and the circle with the dot. This indicates that it is the ***Active Component***.



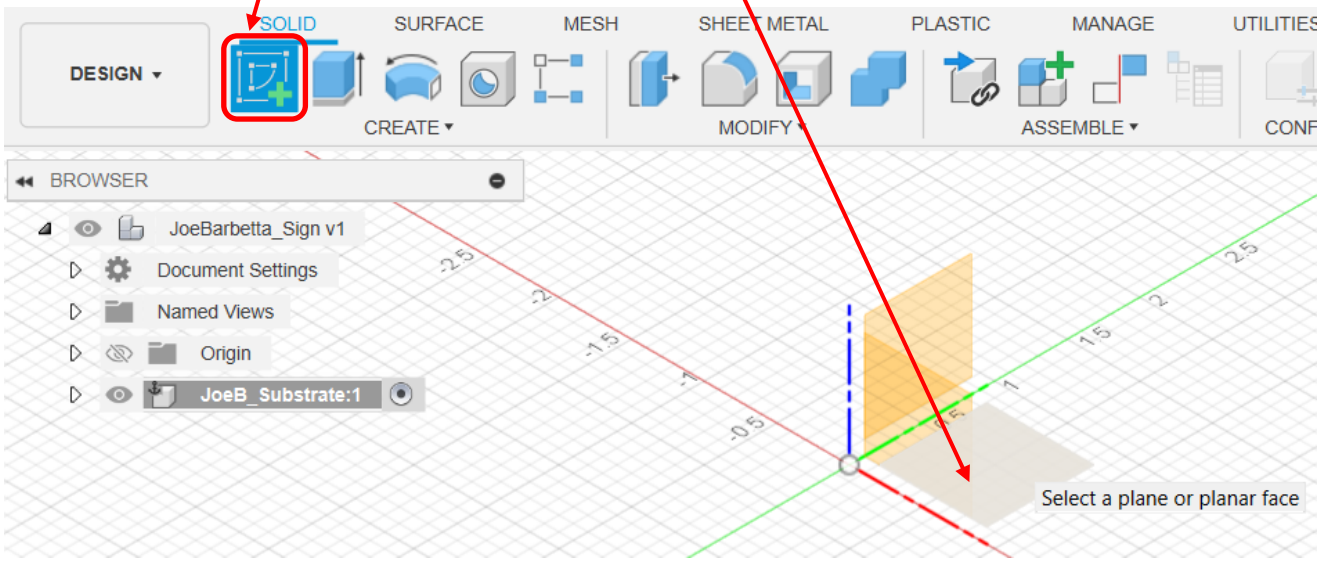

# One reason to hate Fusion!

When a project is saved and then reopened, the Component that was Active is no longer Active.
If one wants to continue working on a Component, it must be reactivated by **right-clicking on the Component Name** and selecting **Activate**. One can also click on the circle to the right of the Component name.

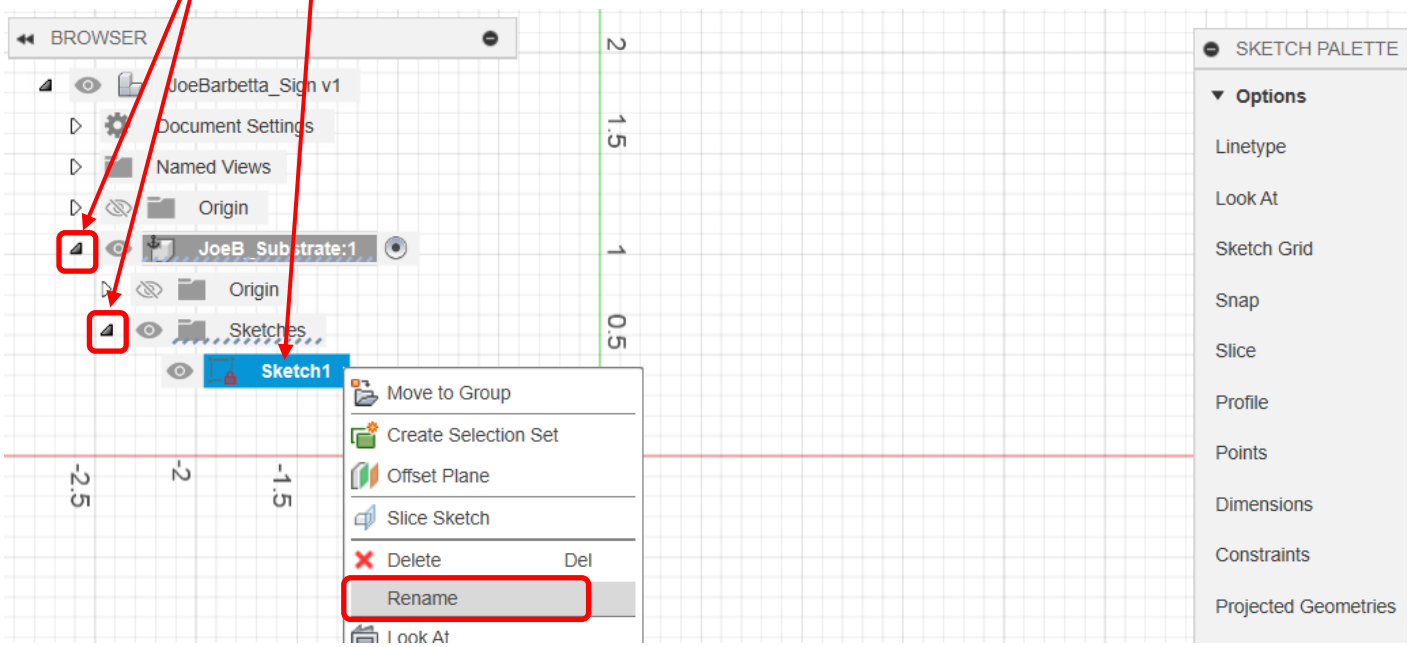


# Creating the First Sketch

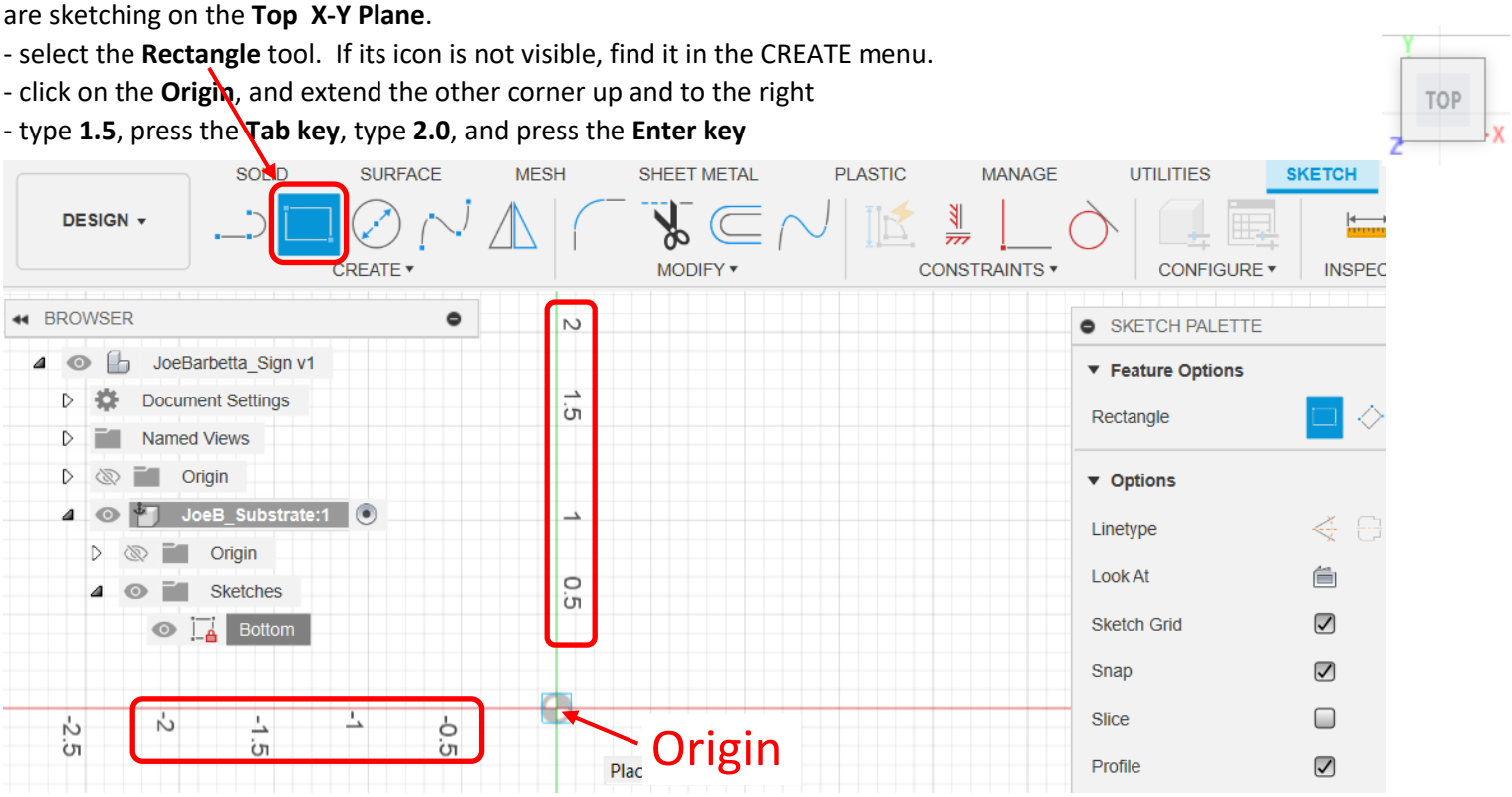
- select the top **Create Sketch** tool and click on the **bottom rhombus** to select the X-Y Plane.
If a tool can't be found, one can always look in the **CREATE** and **MODIFY** menus for it.




- click on the **arrows** next to the Component and Sketches folder to open them
- right click on the **Sketch name**, select **Rename** from the menu and rename the Sketch to **Bottom**

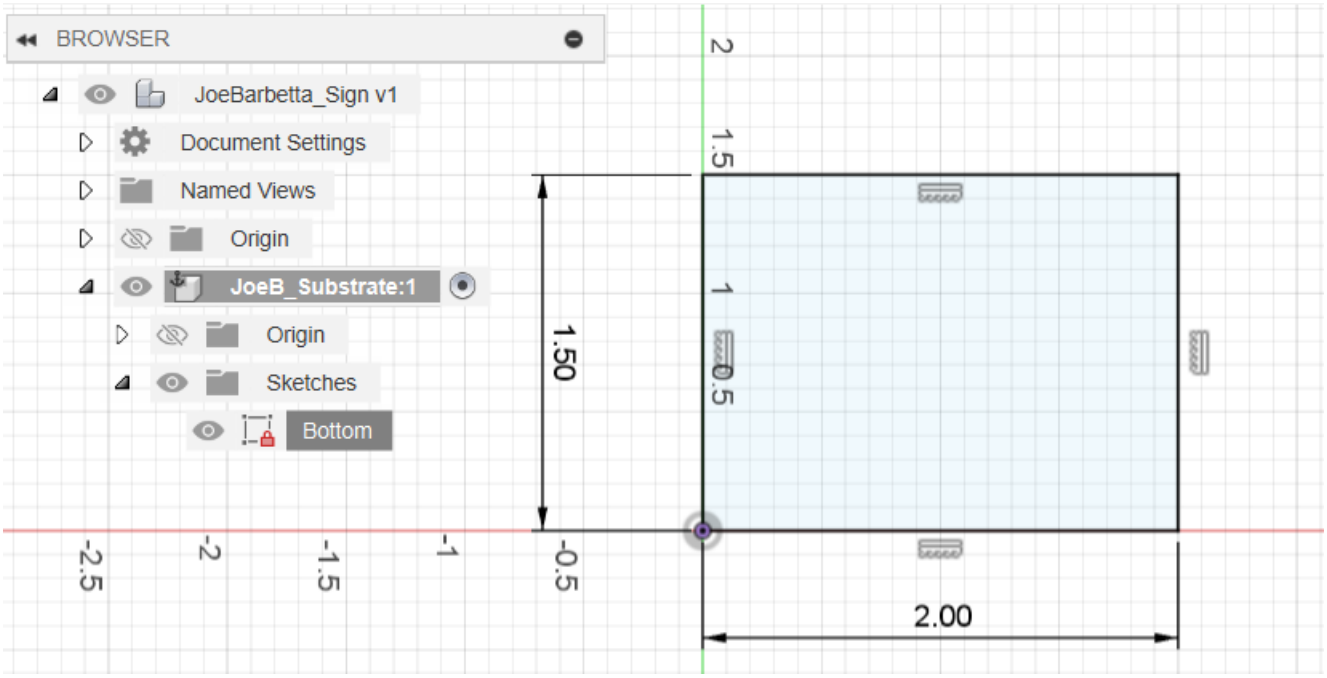


- zoom in as shown below. The scale labels can give an idea of how far one is zoomed in. The **View Cube** should indicate you are sketching on the **Top X-Y Plane**.
- select the **Rectangle** tool. If its icon is not visible, find it in the CREATE menu.
- click on the **Origin**, and extend the other corner up and to the right
- type **1.5**, press the **Tab key**, type **2.0**, and press the **Enter key**

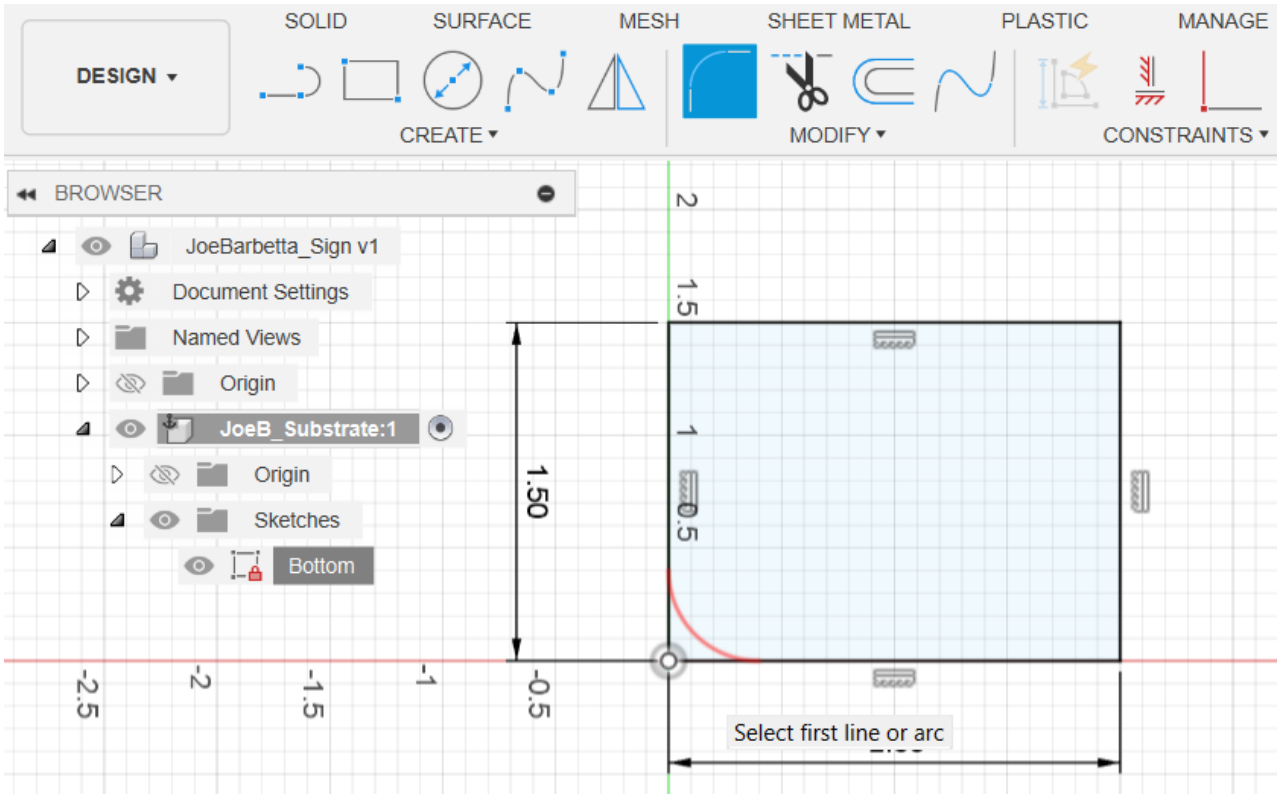



This should be the result. The thin line extending away from the rectangle and that with the arrows and values are Dimension Lines. It is OK if these lines have different positions.

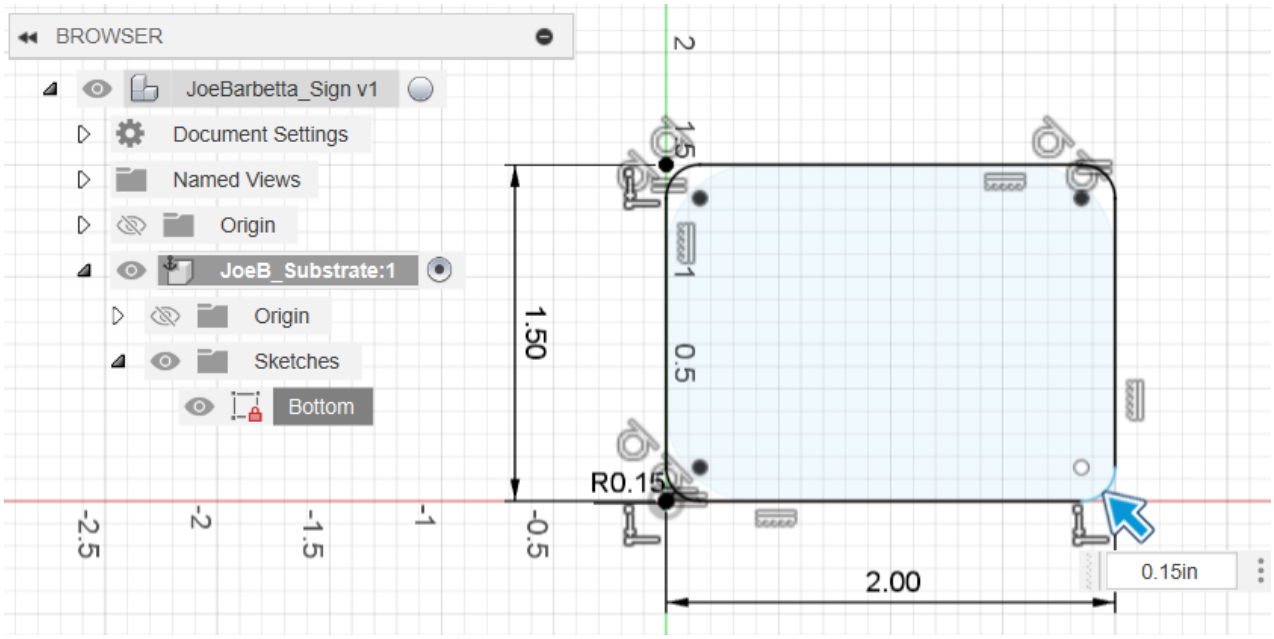
If one ever needs to change a dimension, one can double-click on a Dimension Value and change it.



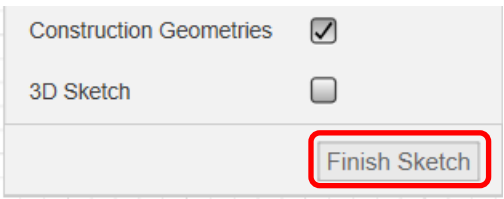
- select the Fillet tool.  If its icon is not visible, find it in the MODIFY menu.
- click on the corner at the Origin and then the other 3 corners



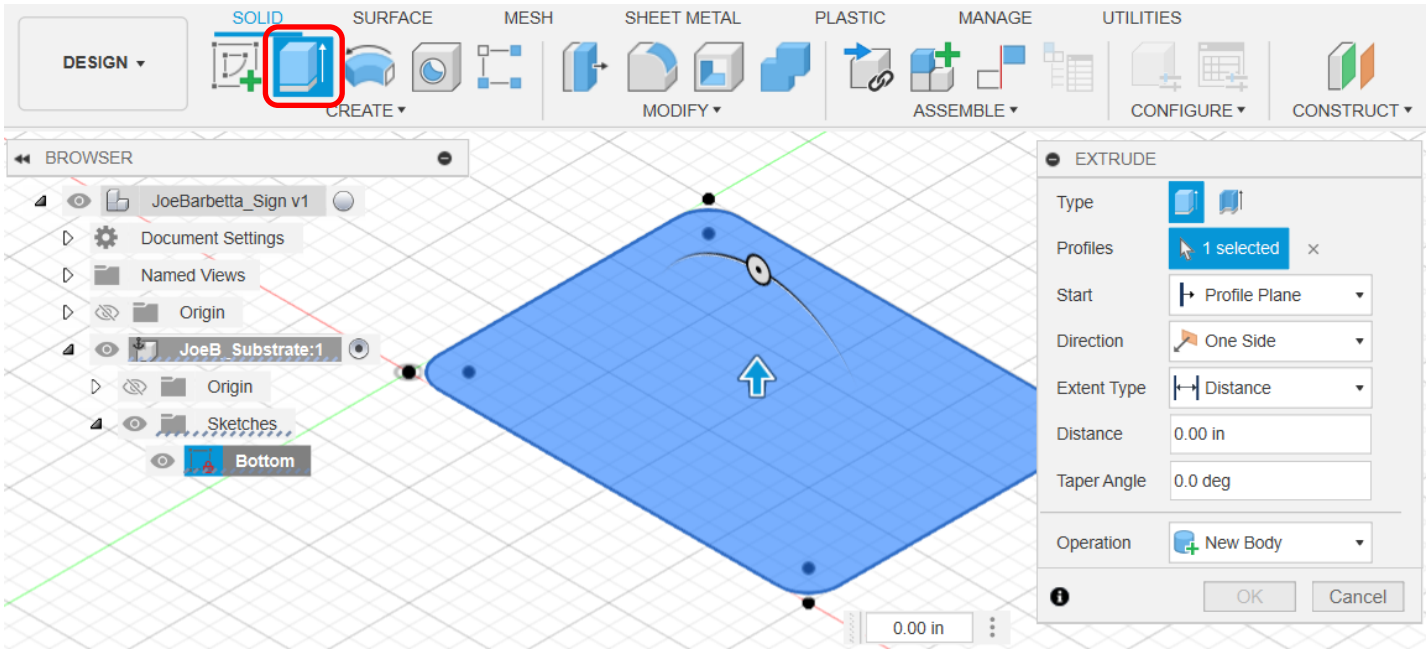

- type 0.15 and press the Enter key.  Any warning message can be ignored.



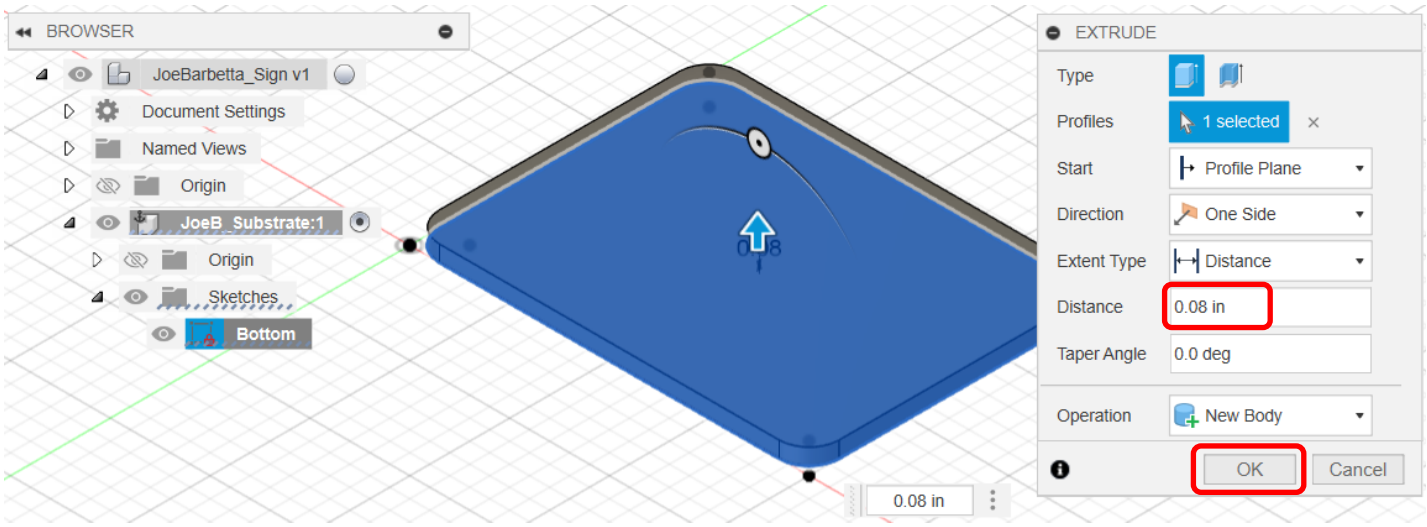

- click **Finish Sketch** at the bottom of the SKETCH PALETTE window



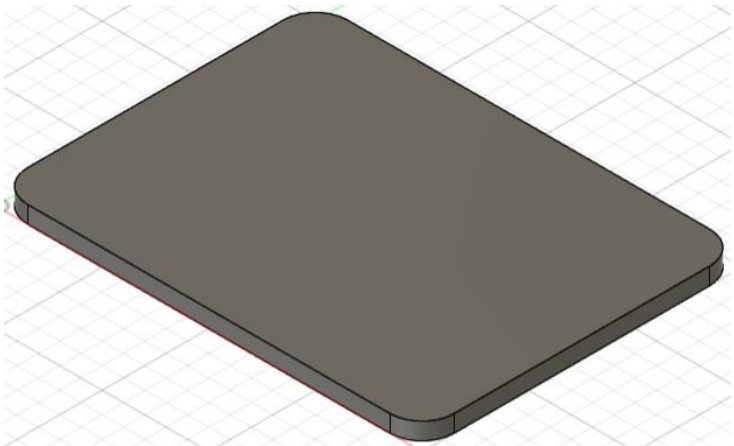
- click on the **Home** icon at the **View Cube**
- select the **Extrude** tool.  If its icon is not visible, find it in the **CREATE** menu.




- for **Distance** enter **0.08** and click **OK**

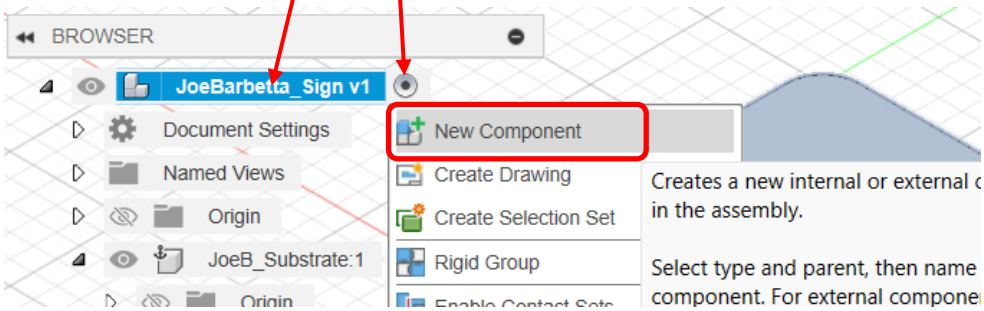



This should be the result.

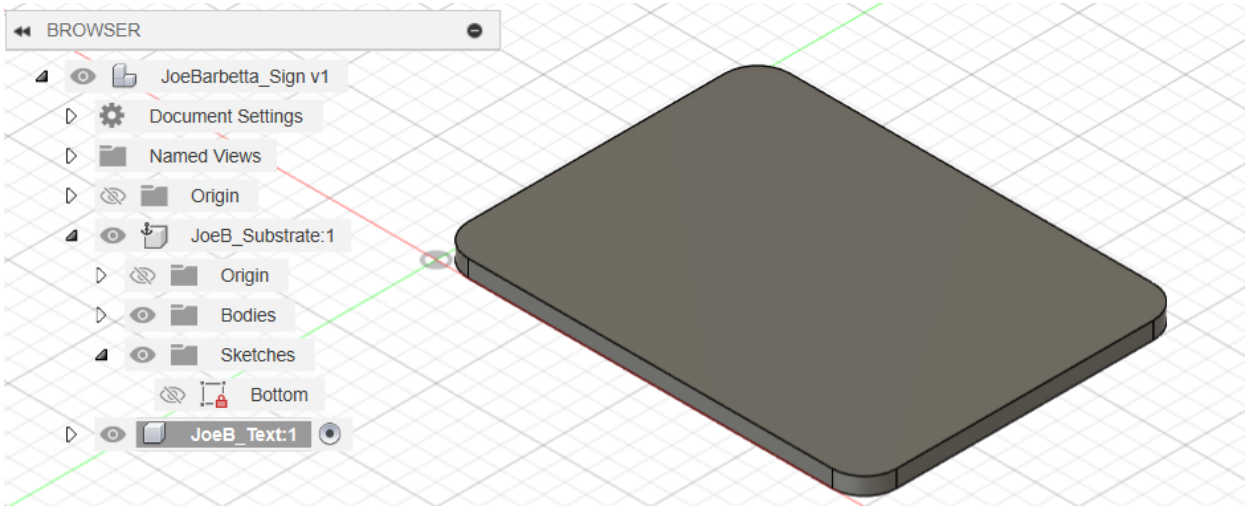


# Creating the Component for Text

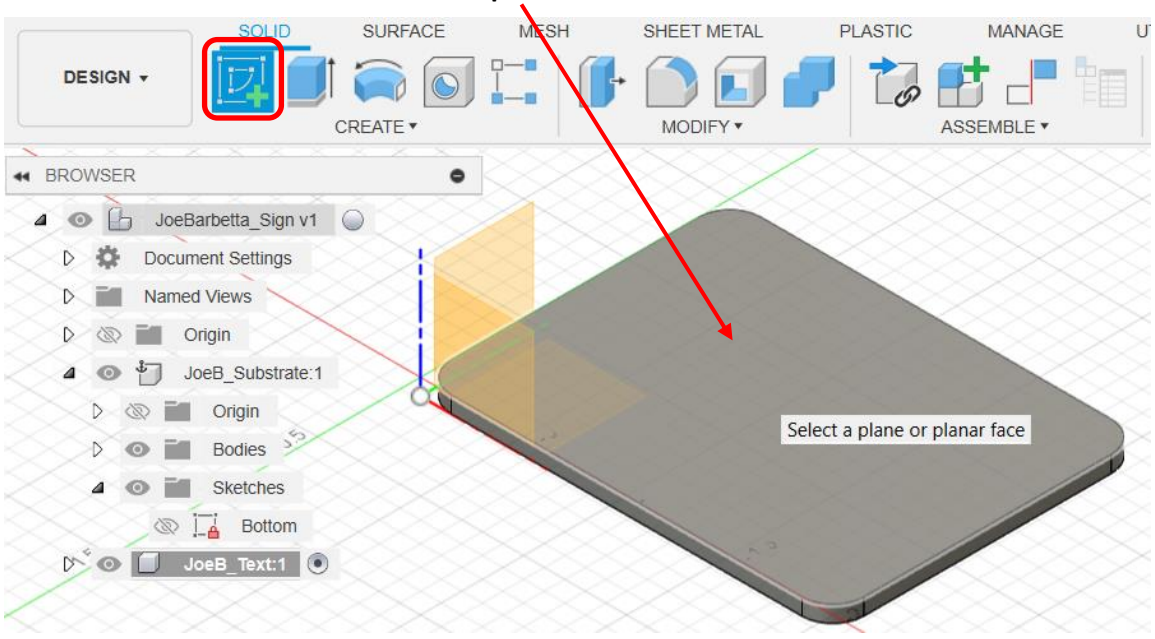
- click on the circle for the **Project Name**. Note that using this circle is an alternative to right-clicking and selecting Activate.
- right-click on the **Project Name** at the top of the BROWSER and select **New Component**
- set **Name** as **Your first name and last name initial followed by _Text** e.g. **JoeB_Text** and click **OK**



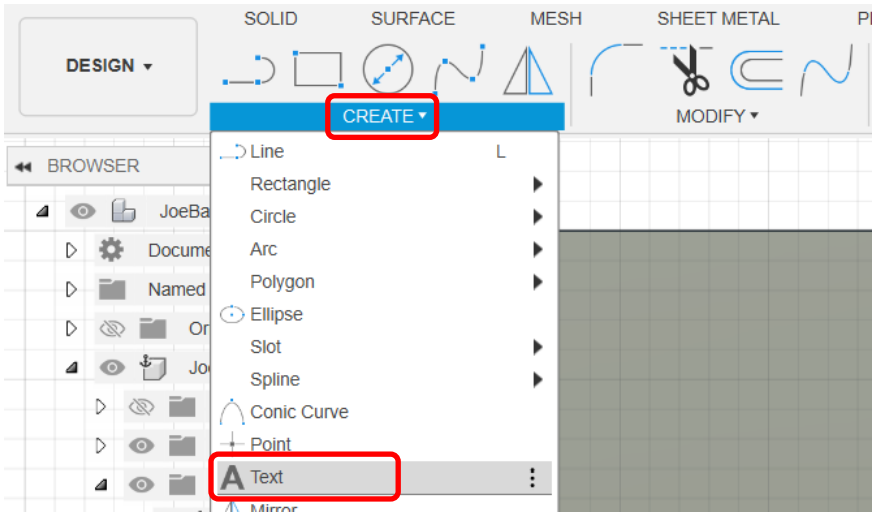

At this point the Substrate body may be dark, as shown below, or it may be "ghosted". This depends on settings in Fusion. It doesn't matter if it is ghosted or not. Note that the new Component is automatically Activated when first created.



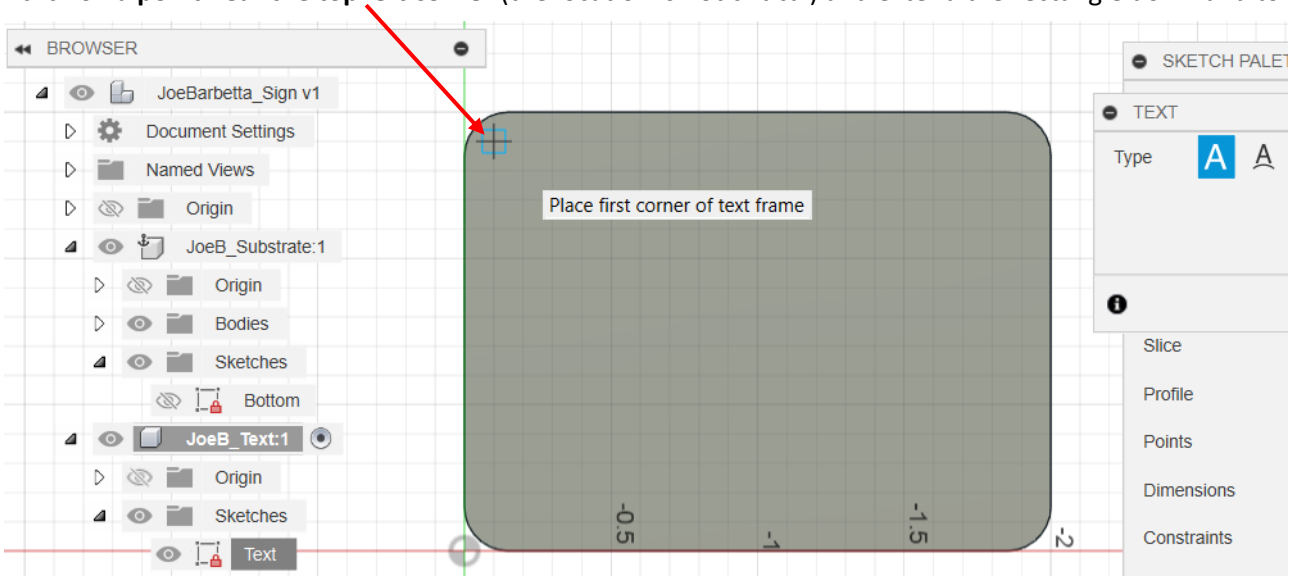

- select **Create Sketch** and click on the **top surface** of the Substrate. Rename the Sketch to **Text**.



- from the **CREATE** menu, select **Text**
- if a **Parametric Text** window appears, click its bottom right **OK** button

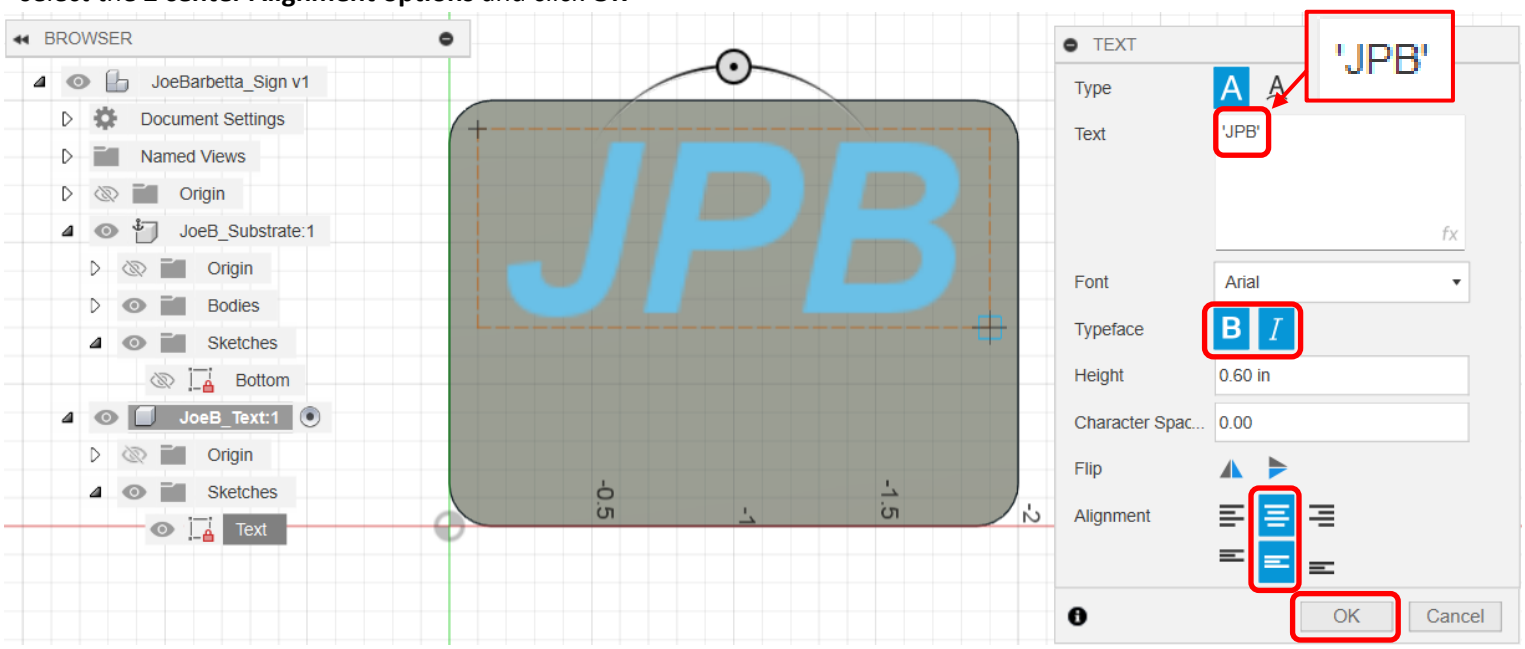



- click on a **point near the top left corner** (the location is not critical) and extend the rectangle down and to the right




- click on a point about **1/2 the way down and near the right edge**. The location is not critical.

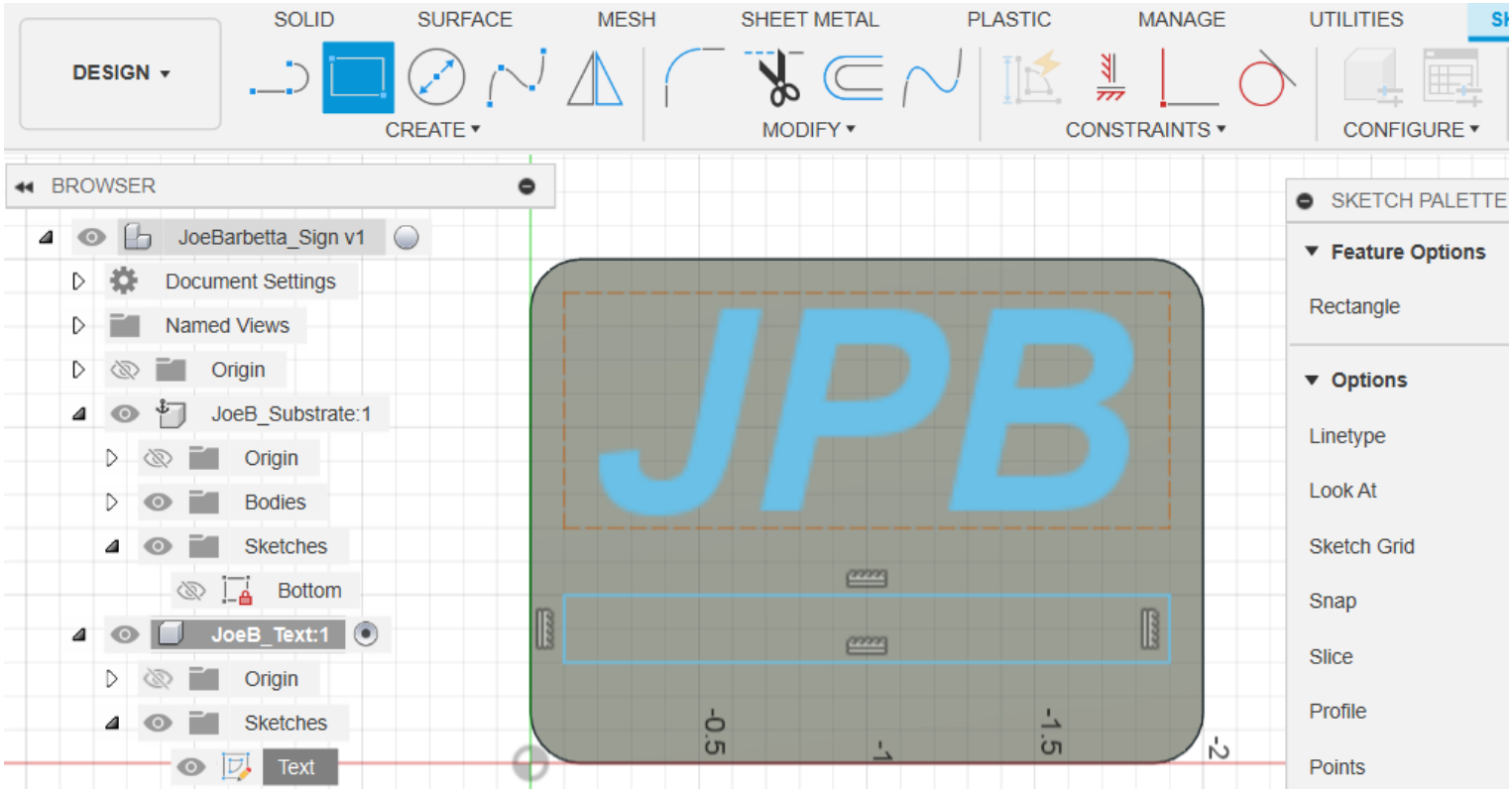
- in the Text box enter your **3 initials preceded and followed by a single quote, e.g. ‘JPB’**
- click on the **Bold** and **Italic** icons to highlight them blue
- set the **Height** to **0.60**
- select the **2 center Alignment options** and click **OK**



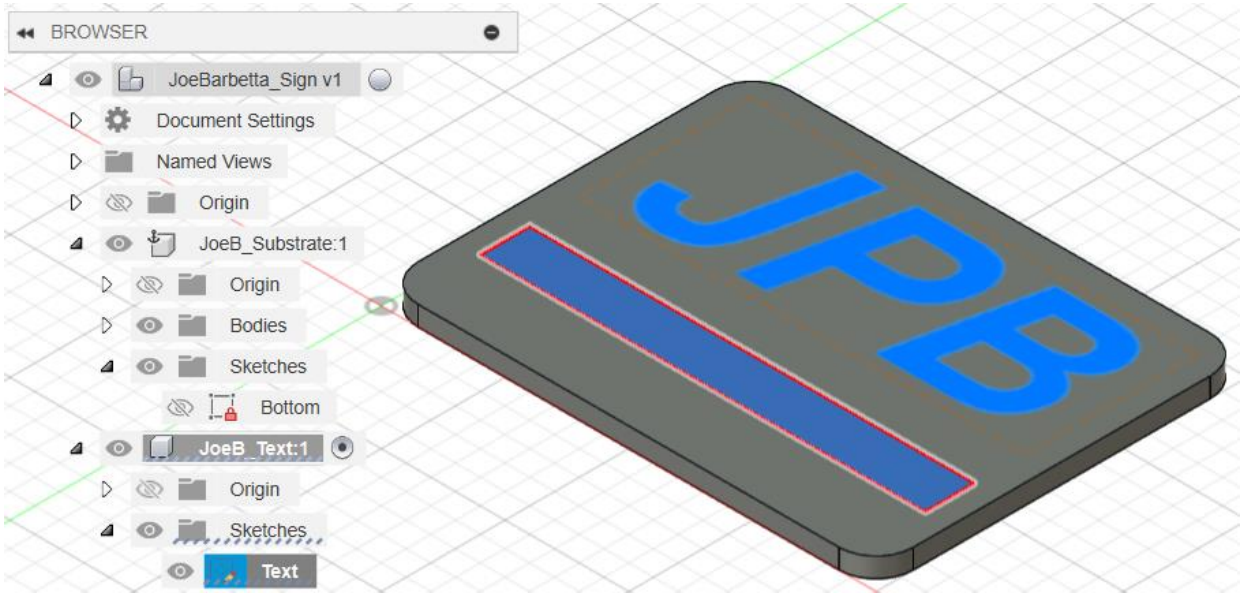

- under your initials create a design using the Line, Circle, Spline tool, etc. Here a simple rectangle was created to demonstrate later steps, but you can do much better. Time to earn that art credit.

- if desired, one can right-click on the text and select **Edit Text** and change the Height of the text or any other text parameters

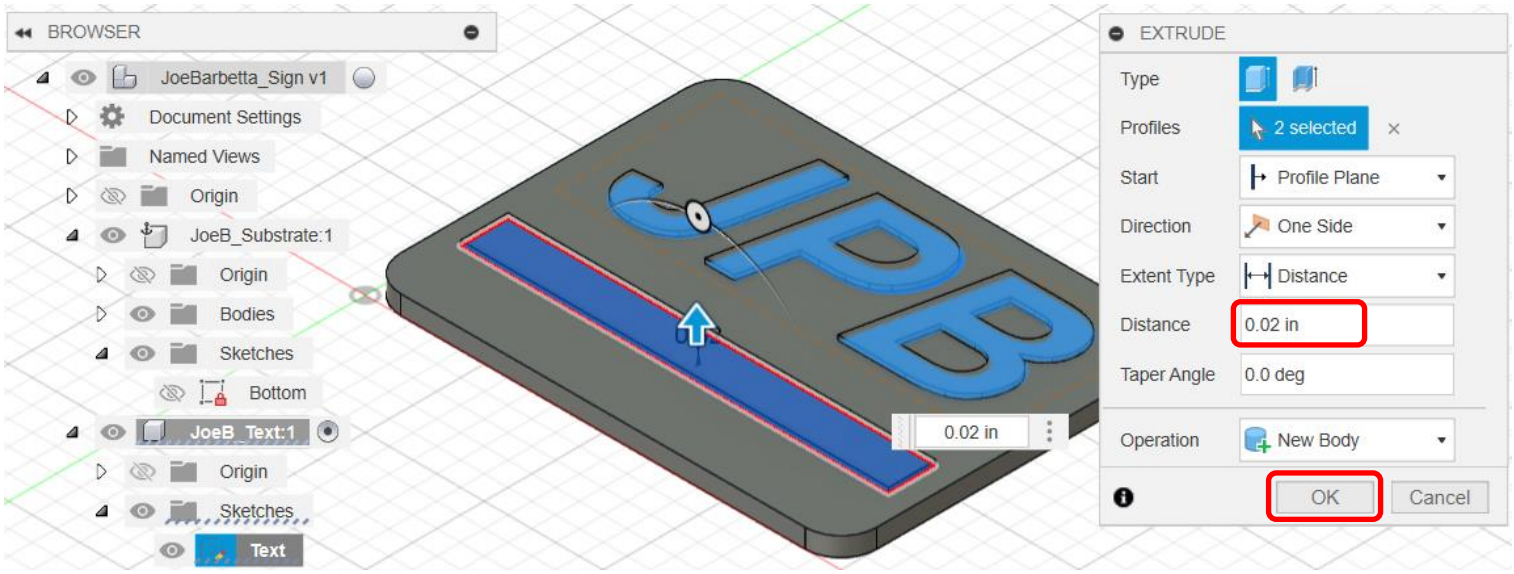
- when done, click **Finish Sketch**



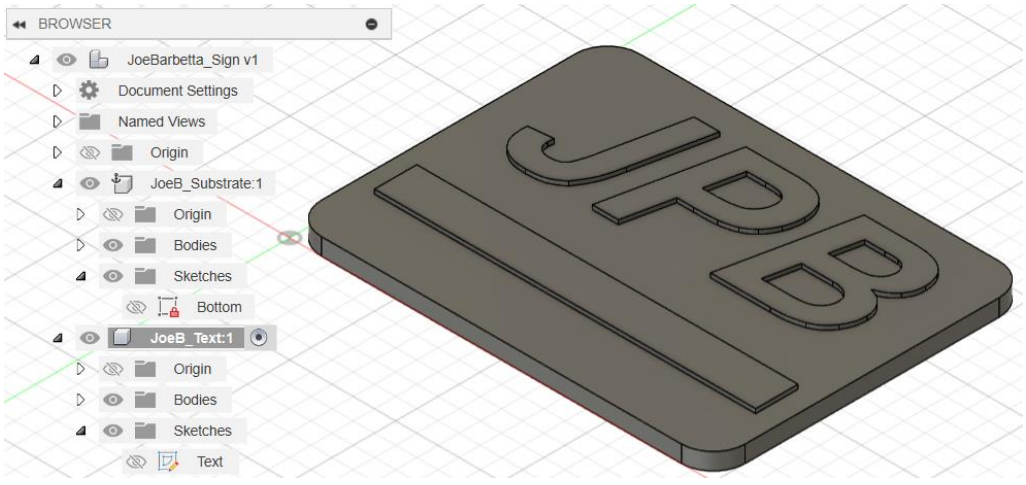
- hold down the **Shift key** and **click on your initials** (or other text) and then the **bottom shape(s)** to cause all of the to turn blue



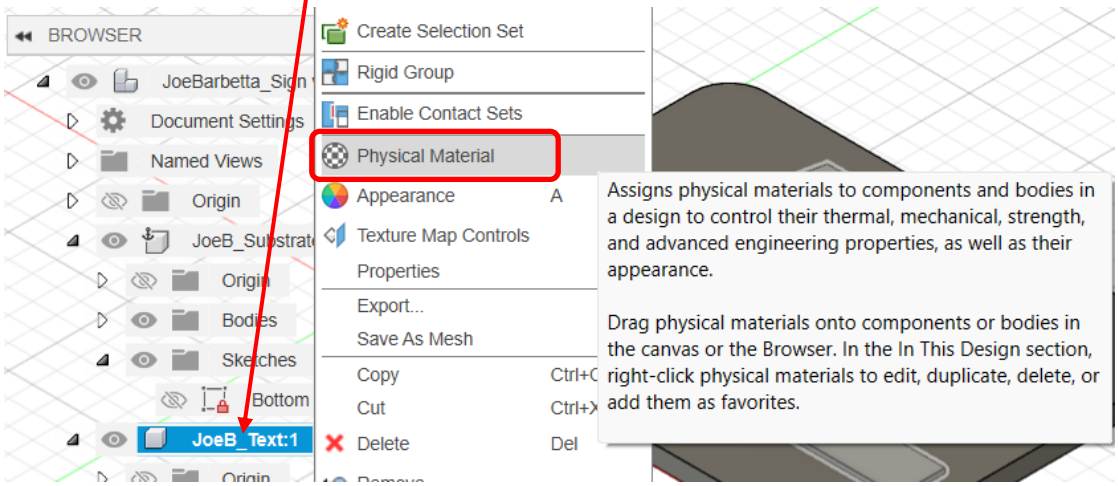

- select the **Extrude** tool
- for **Distance** enter **0.02** and click **OK**




The result should look like that below.  Of course with you customizations.

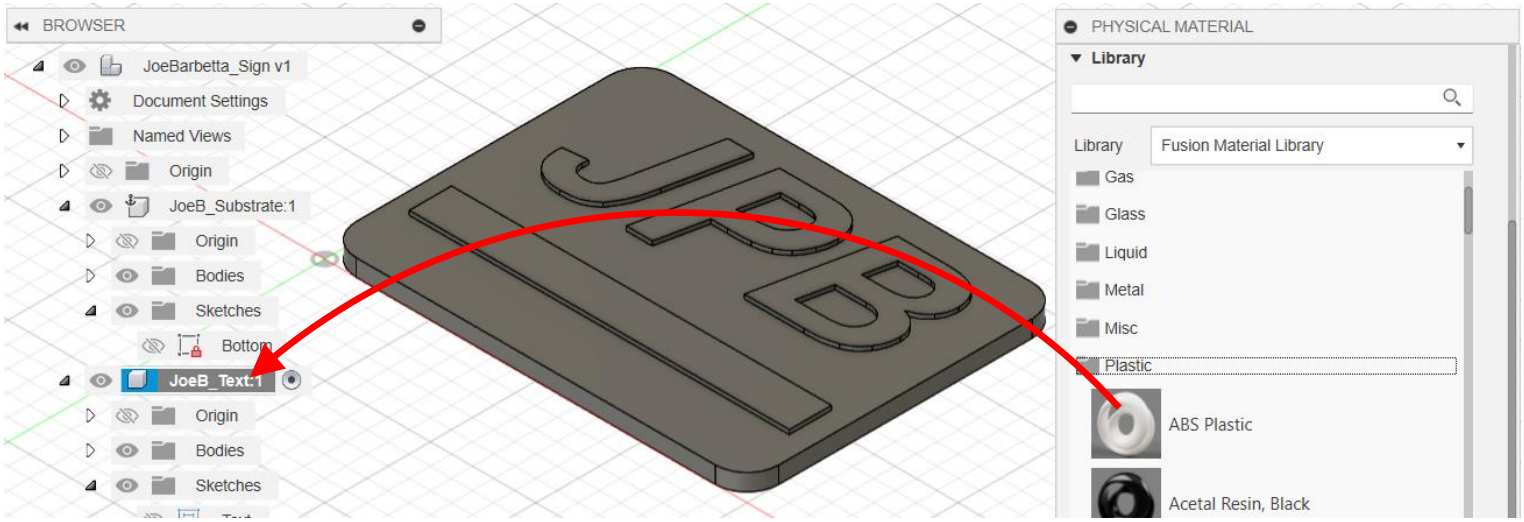


- right-click on the **Text Component** and select **Physical Material**

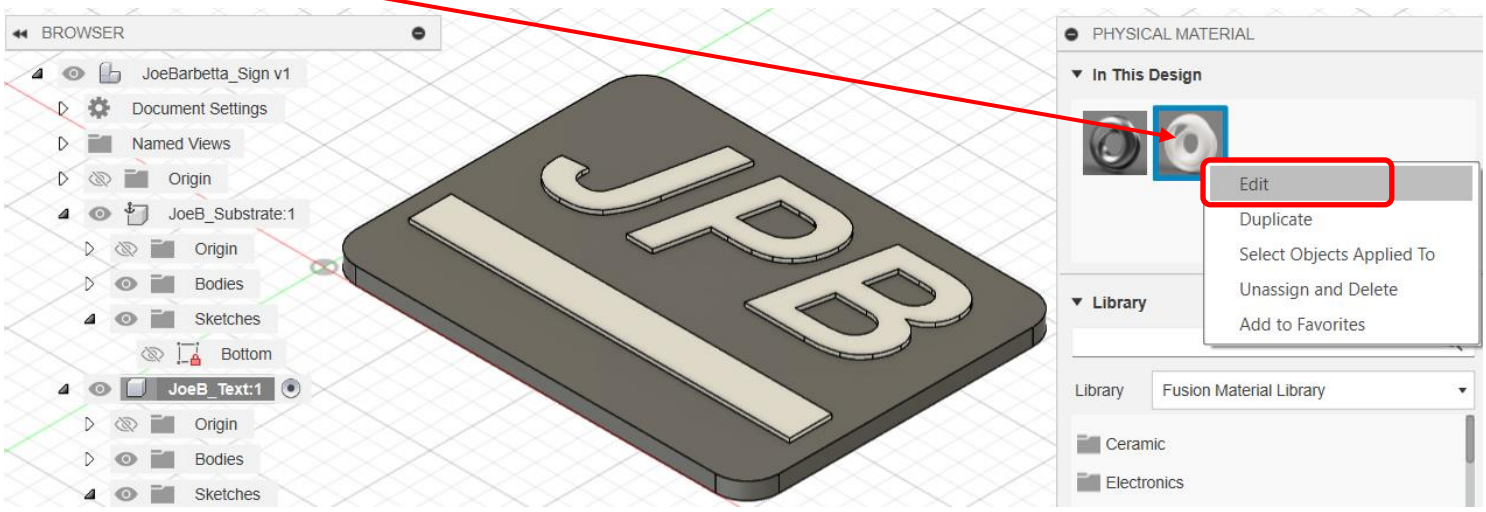



- scroll down to the **Plastic** folder and click on it
- drag the **ABS Plastic** icon onto the **Text Component**

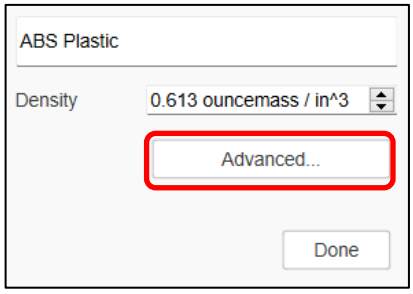
As of this writing, PLA, which is a more common 3D printing material, does not appear in the list. It actually doesn't matter what material is selected in Fusion. The filament loaded into the printer determines the material and color. We are applying a plastic and then changing its color just for aesthetics in Fusion.



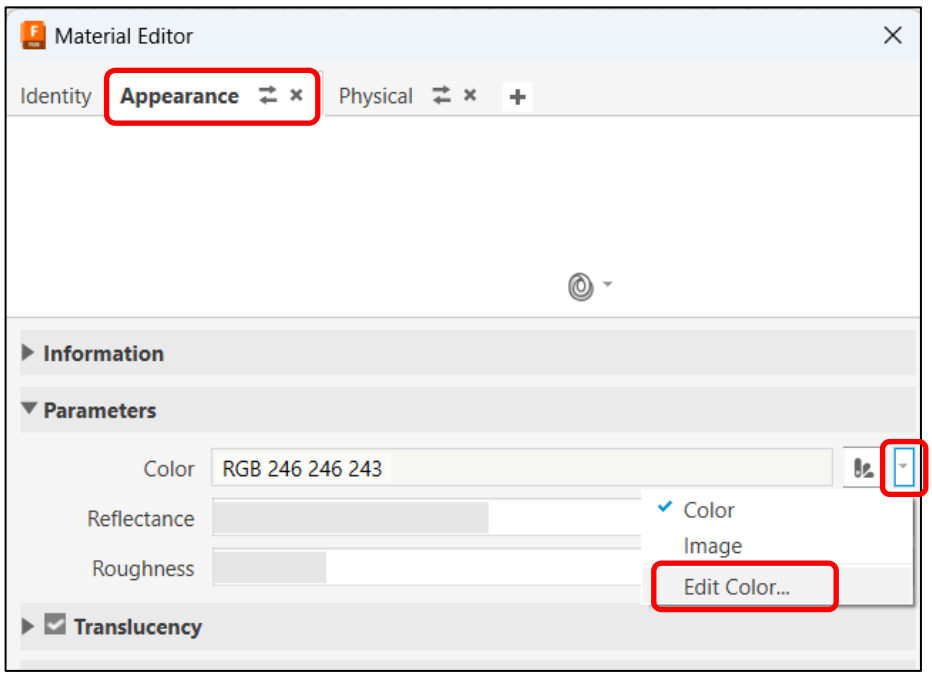

- right-click on the **ABS Plastic** icon and select **Edit**



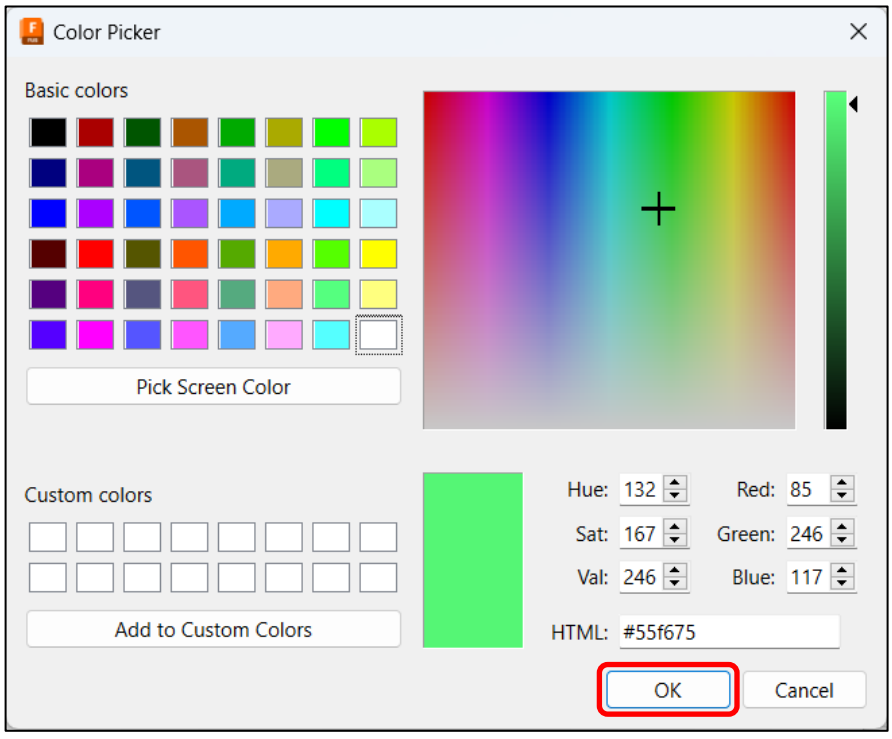
- click on Advanced



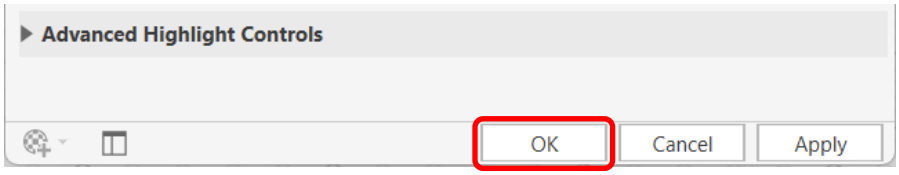

- select the **Appearance** tab and click on the **drop-down menu button** and select **Edit Color…**



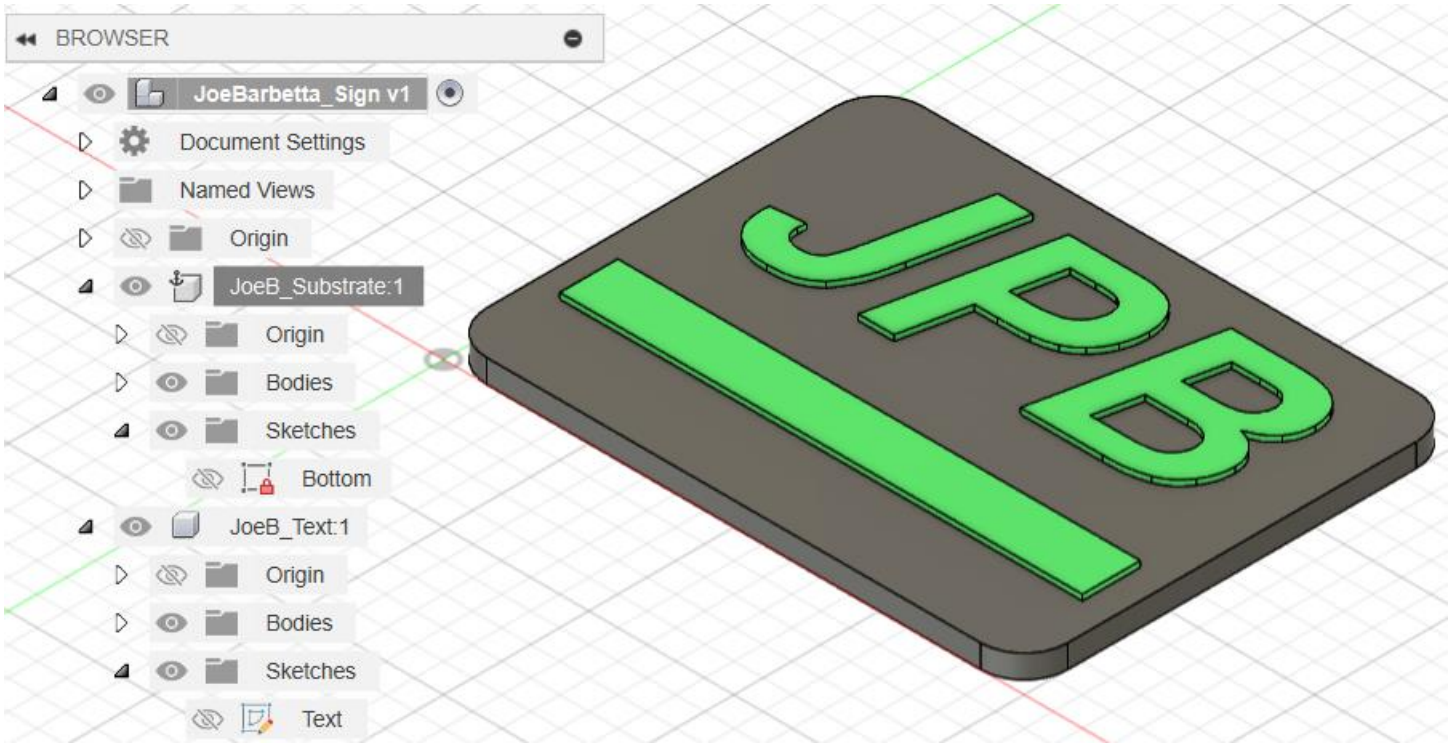

- select a color by either clicking on a **Basic colors** box or **color spectrum** box. The brightness can be adjusted with the right-side slider. Click **OK**.



- click **OK** at the bottom of the Physical Material window

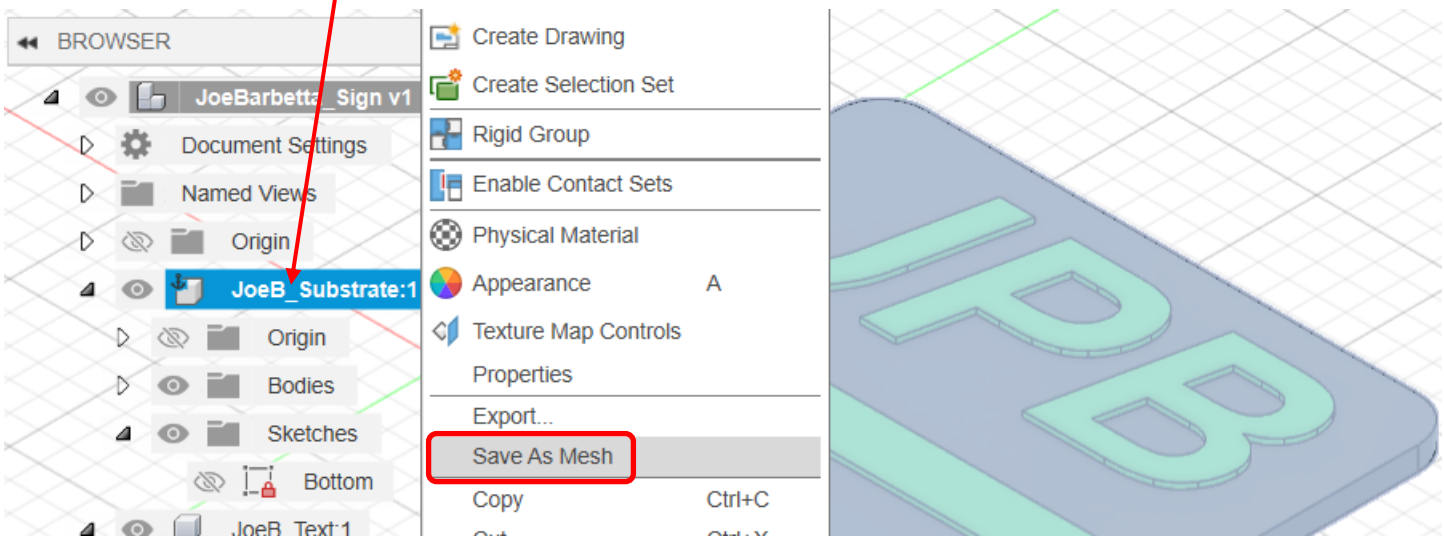



The Text and Art should change color.

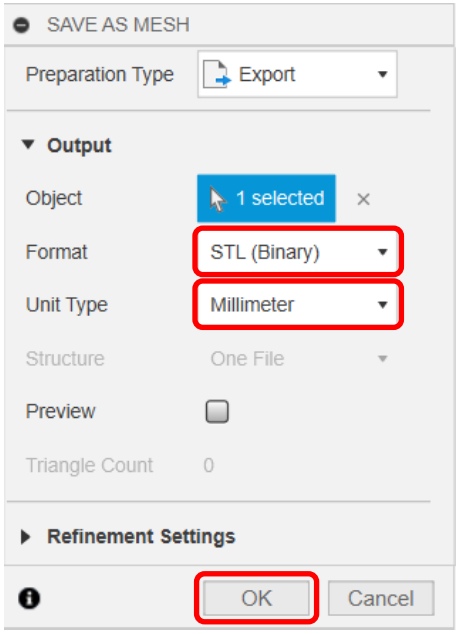



## Exporting STL files

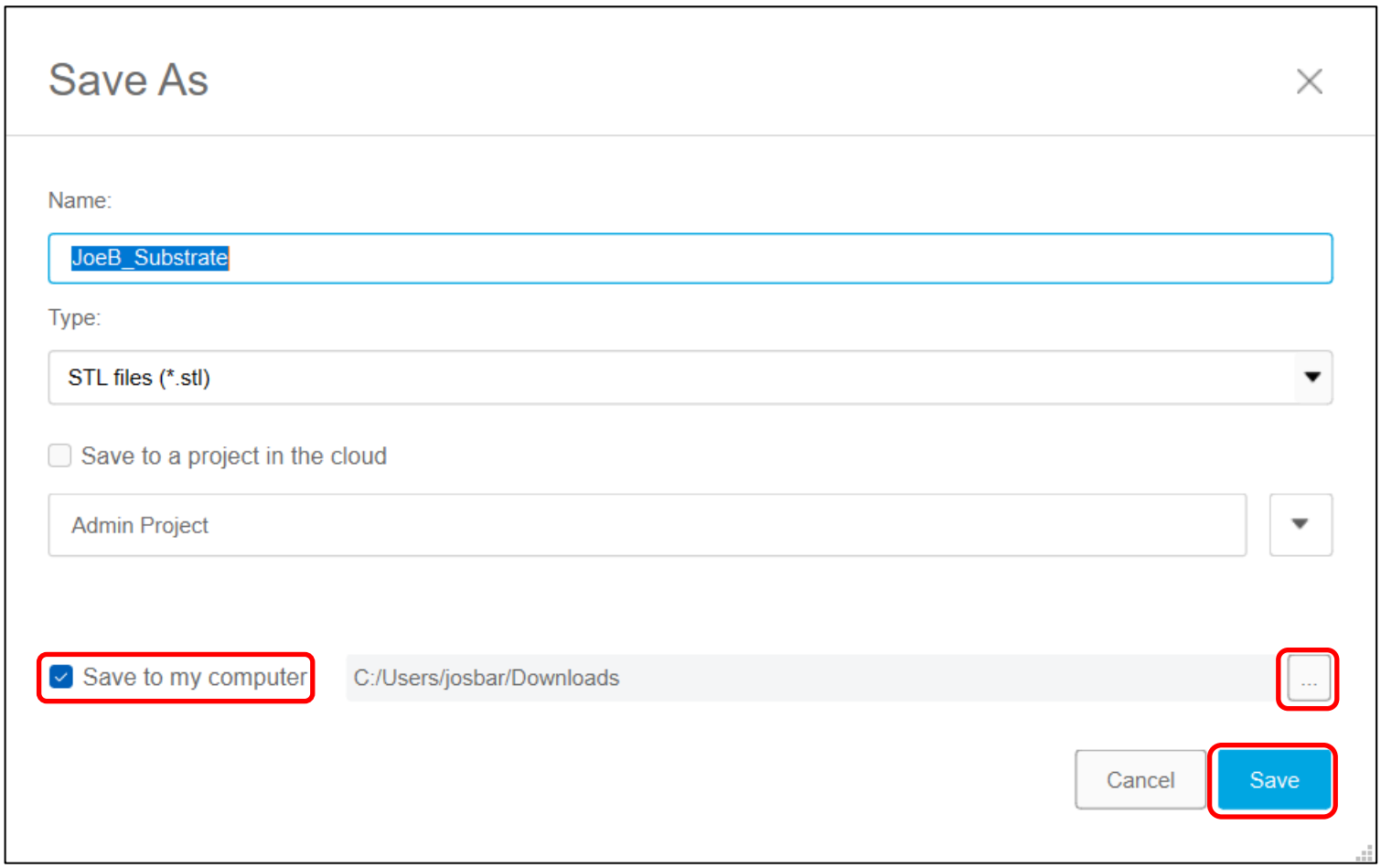
- right-click on the **Substrate Component** and select **Save As Mesh**



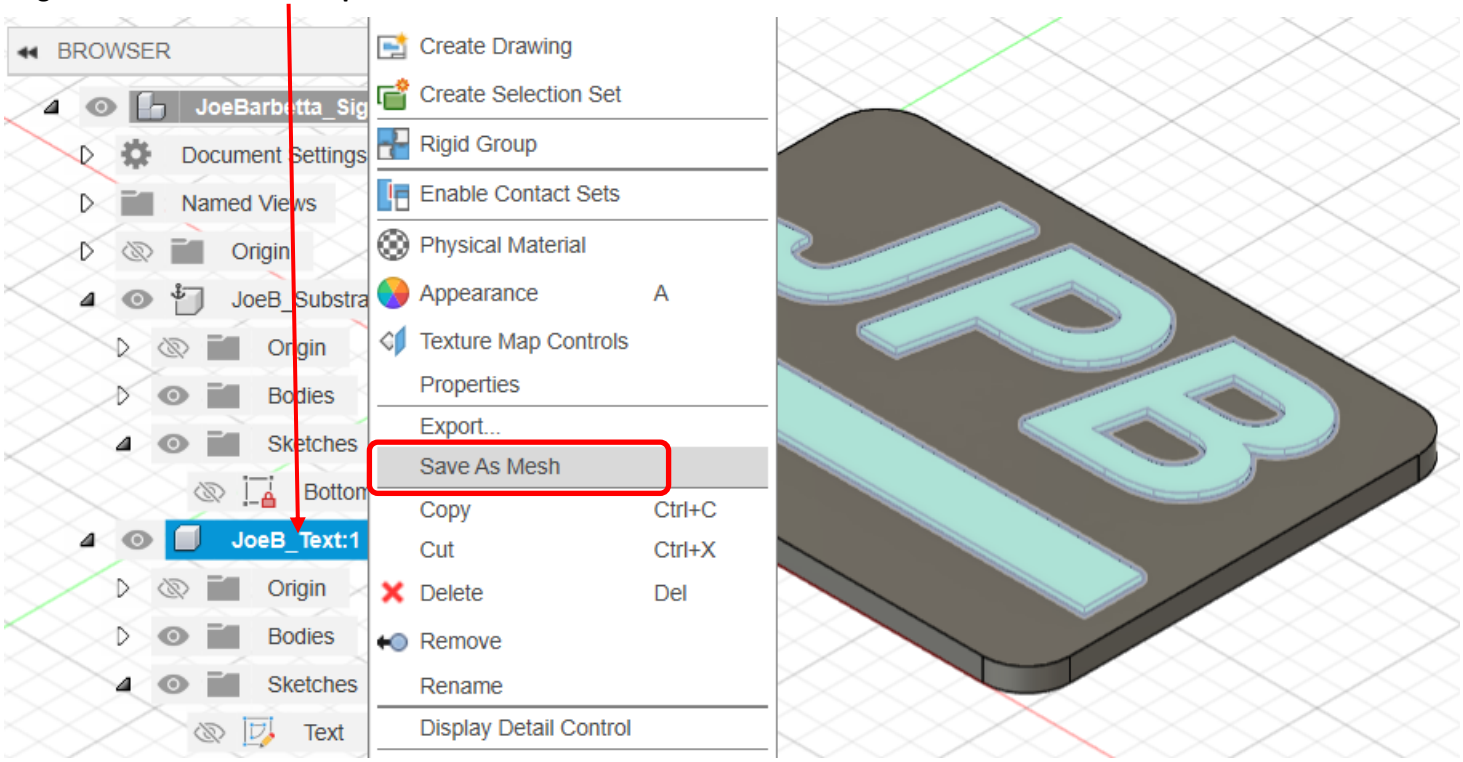
- set **Format** to **STL (Binary)** and **Unit Type** to **Millimeter**




- ensure that **Save to my computer** is checked
- note the **save location**. By default it should be the computer's **Downloads** folder. If desired, one can click the button with the 3 dots to change the save location.

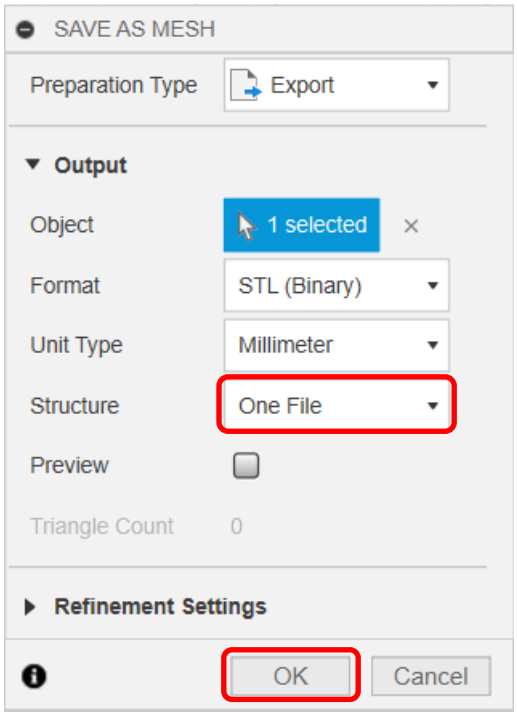


- right-click on the **Text Component** and select **Save As Mesh**




- change **Structure** to **One File** and click **OK**

Note that the Structure option was not available when a STL was created for the Substrate Component. It is now an option because there are multiple bodies in the Component for the different letters and art shape(s). The alternate Structure option is **One Body Per File**, which would result in a STL file being created for each body. In this case we only want a single STL file for the text and art shape(s).



# Using Cura – Dual Component

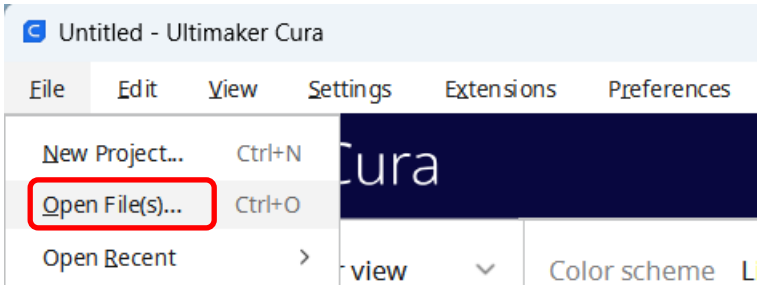
Cura is a free Slicer program offered by Ultimaker, which can be used with many different 3D printers, including those from manufacturers other that Ultimaker. The initial instructions apply to using a printer with a ***dual extruder***. Afterwards, instructions for using a less expensive printer with a **single extruder** will be offered.

A printer with a dual extruder allows two different filament spools of different colors to be loaded at once and the printer will automatically switch between them.

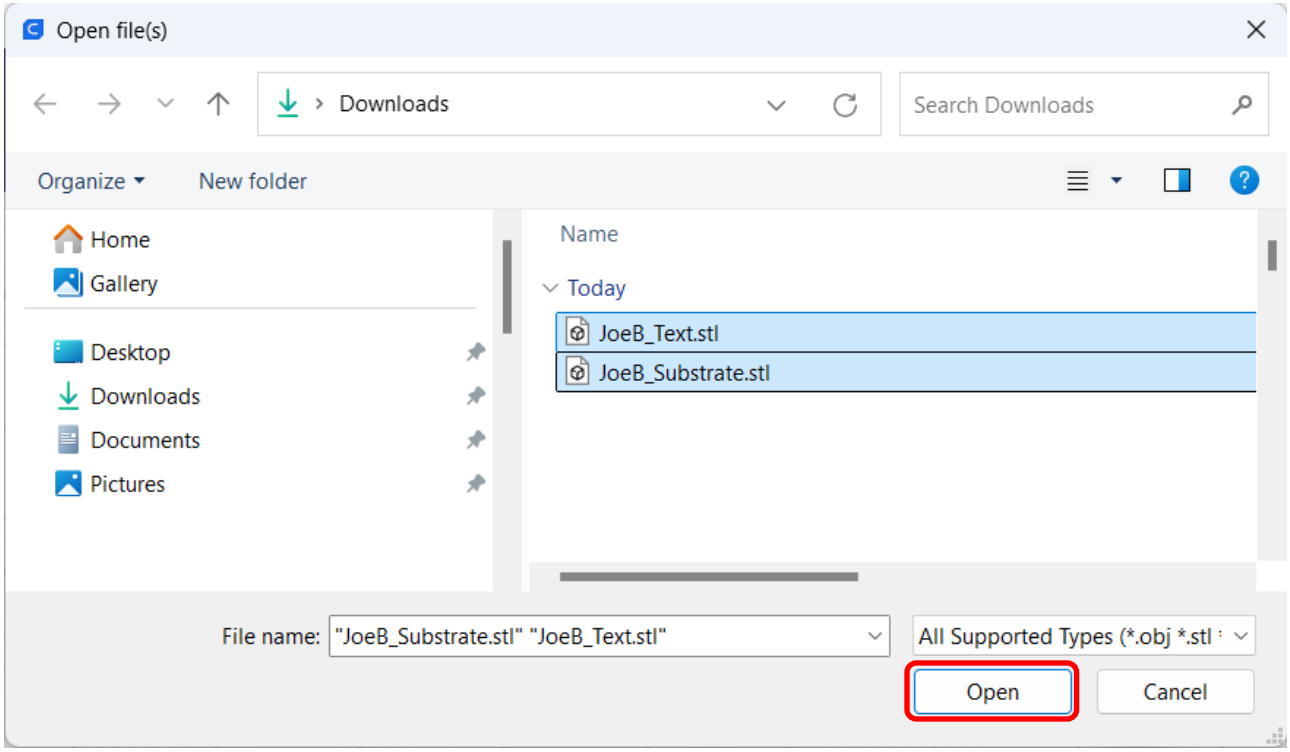
A printer with a single extruder, will require the printing to be paused after the substrate is printed. The filament is then manually changed and primed and then printing is resumed. Note that multiple dual color items can be printed with the need for a single filament change, if the substrates are all the same heights.

This section is for a model that has been separated into **2 STL files**. One for each color. **If there is only 1 STL file**, one can consult the **Using Cura – Single Component** section.

- from the **File** menu, select **Open File(s)…**

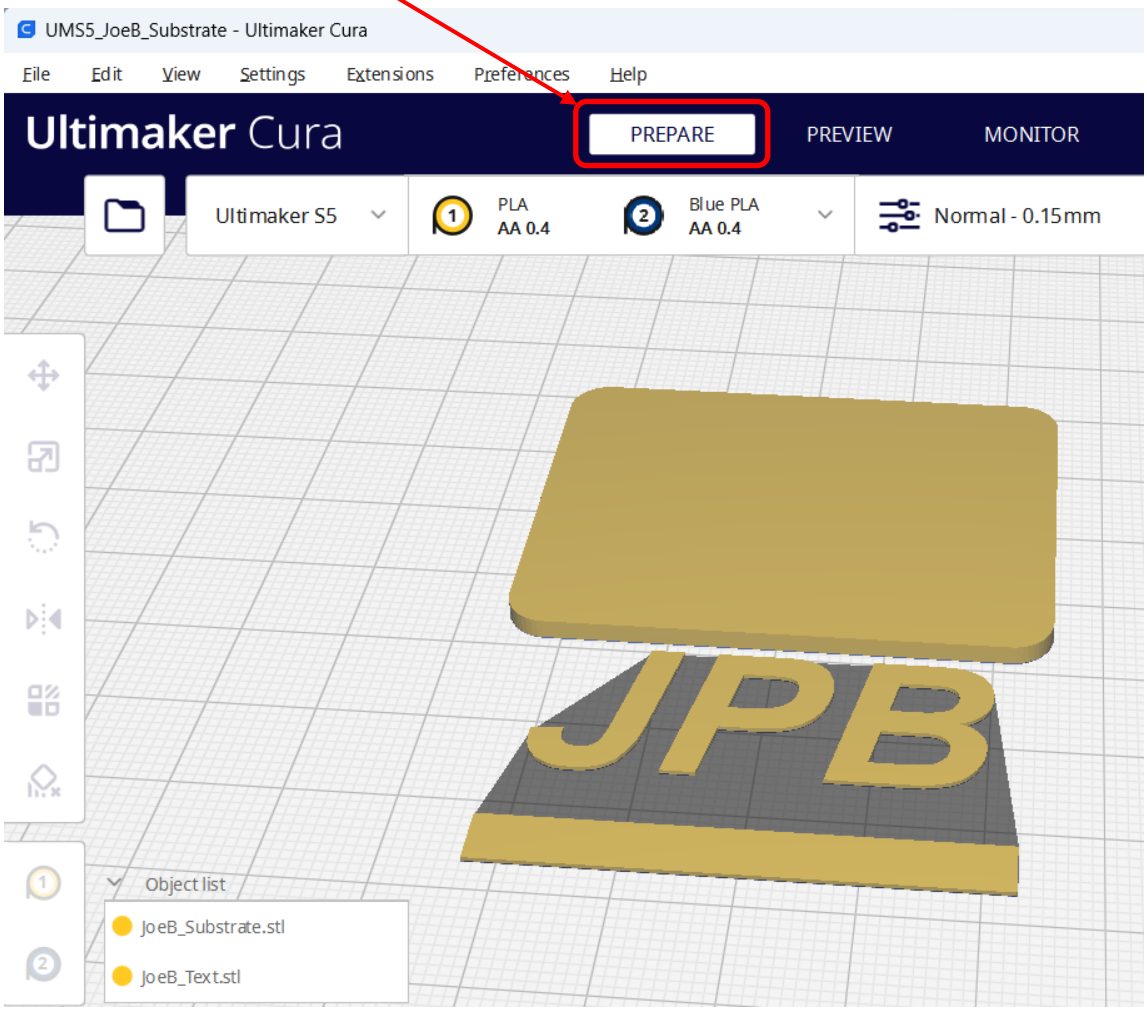



- navigate to the location of the files, which should be the **Downloads** folder and **select both the Text and Substrate files**
- click **Open**

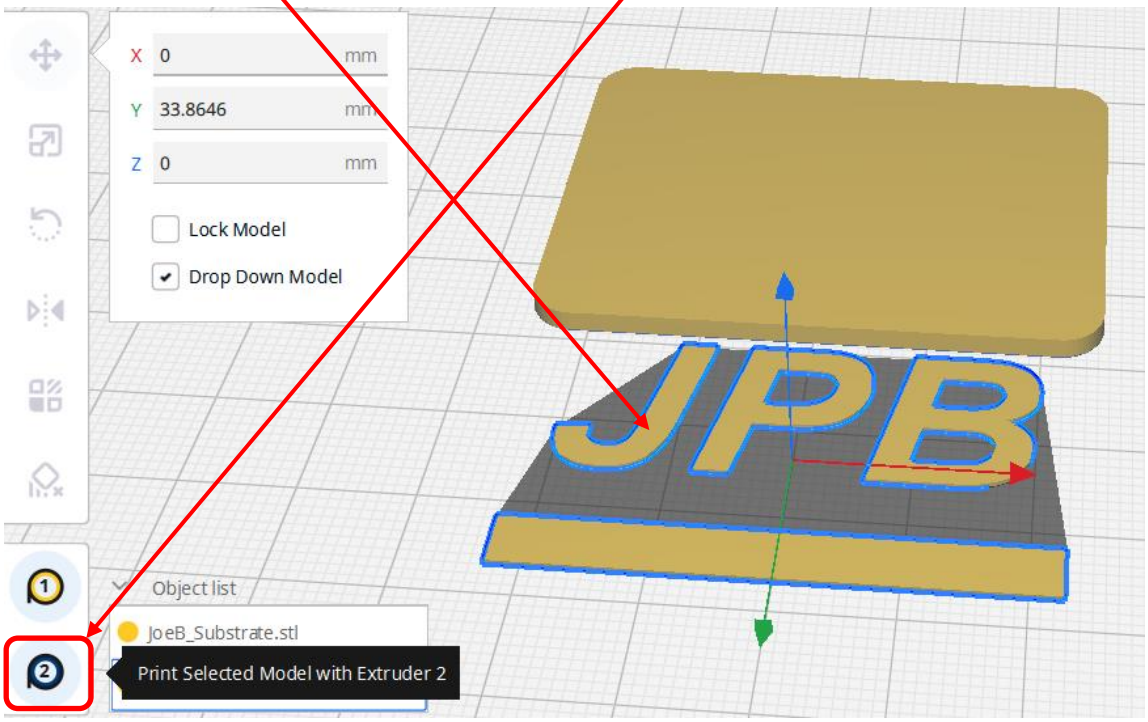


- ensure that the **PREPARE** view is selected

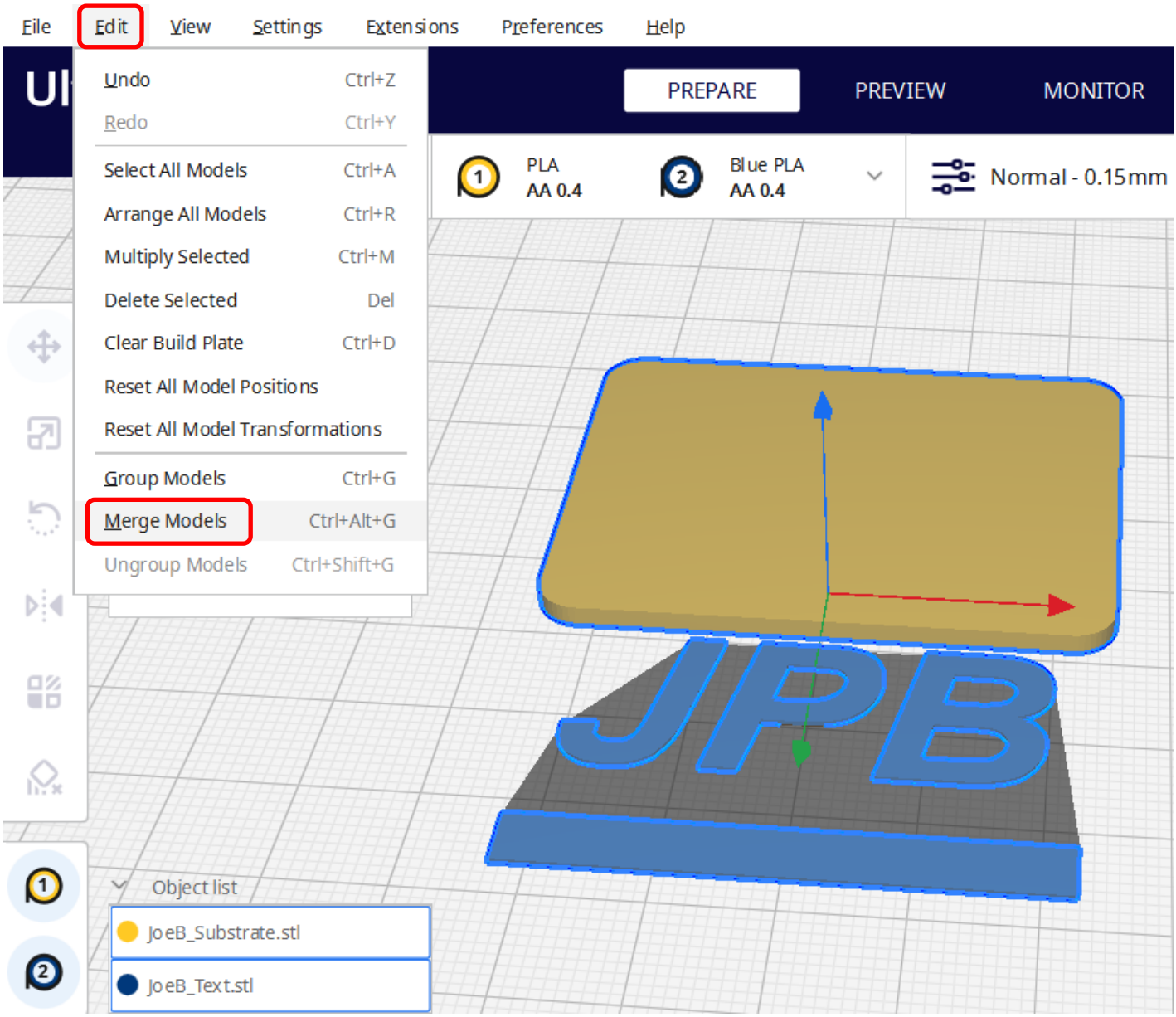
By default Cura should place the two components next to each other on the build plate.



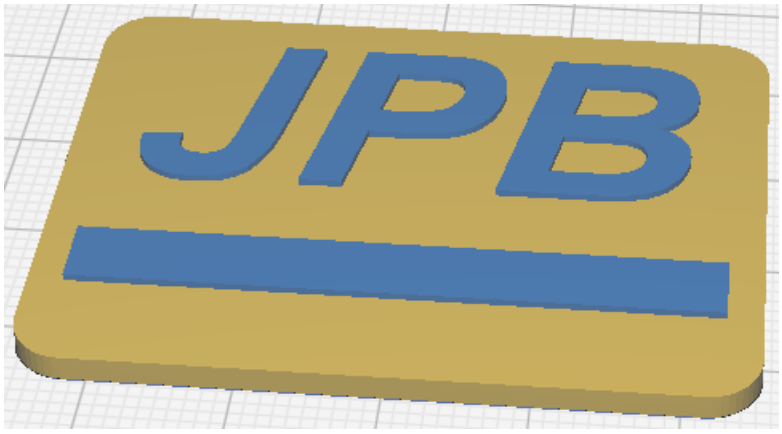

- click on the **Text Component** and then on the **Extruder 2** icon at the bottom left



- hold the **Shift key** and click on the **both the Substrate and Text components**
- from the **Edit** menu, select **Merge Models**

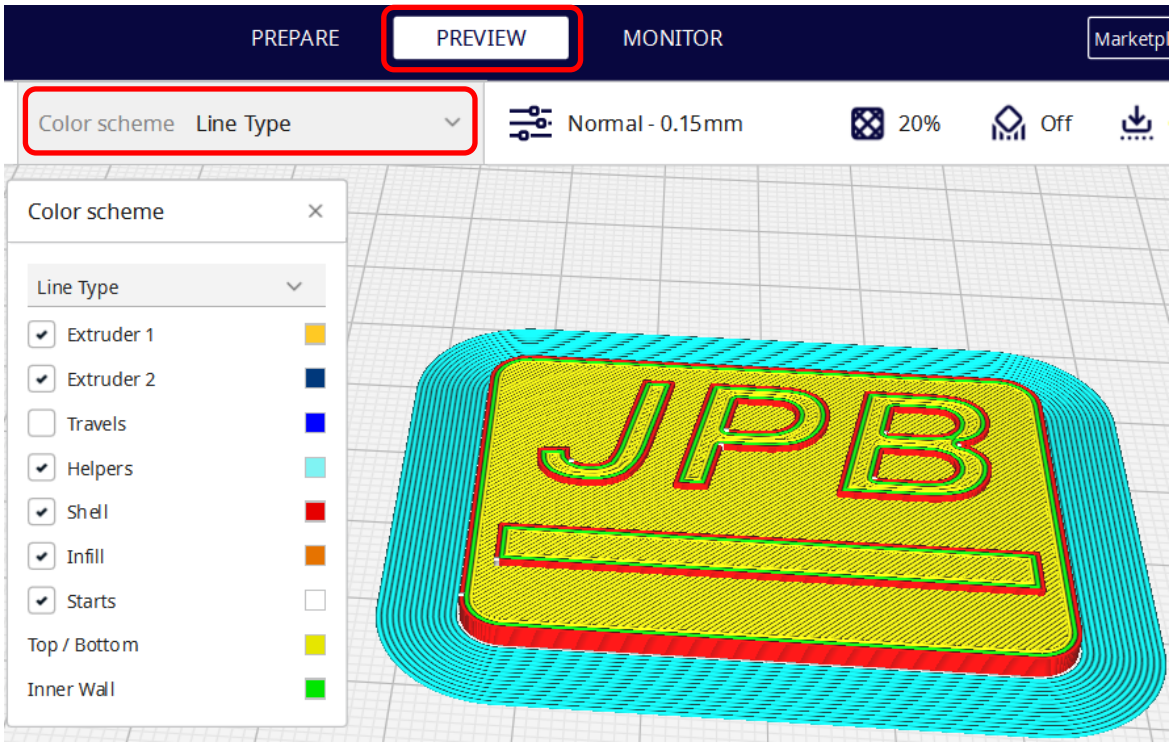



This should be the result of the Merge.

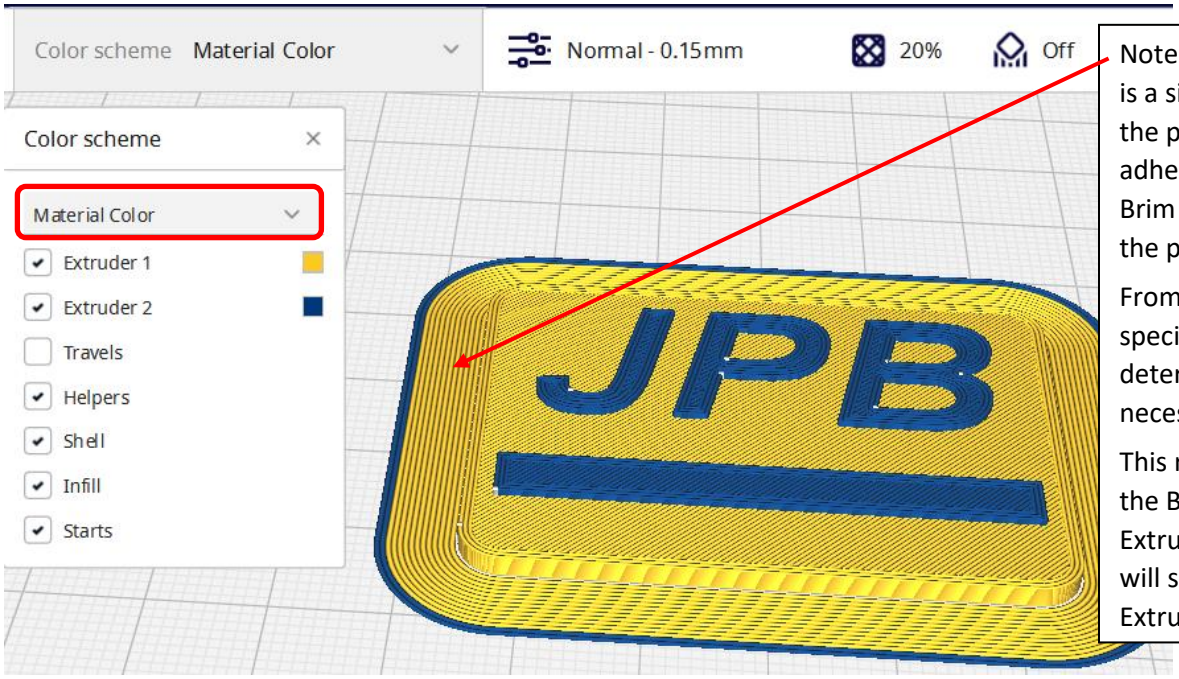


- click the bottom right **Slice** button


- click on the top center **PREVIEW** selection and then on the **Color scheme** options arrow
By default, Cura assigns colors to match Line Types.




- change the **Color scheme** to **Material Color**
Note that the color is assigned according to Extruder number.

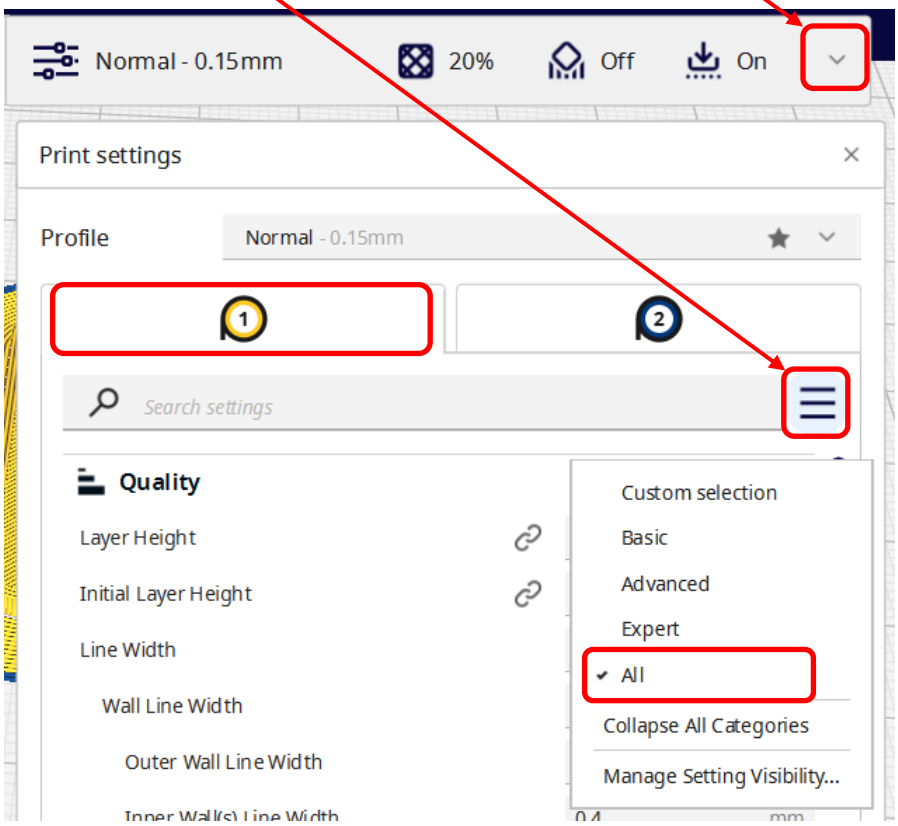



Note the Brim color. The Brim is a single layer printed around the part to help ensure adhesion to the print bed. The Brim is then peeled away when the print is done.

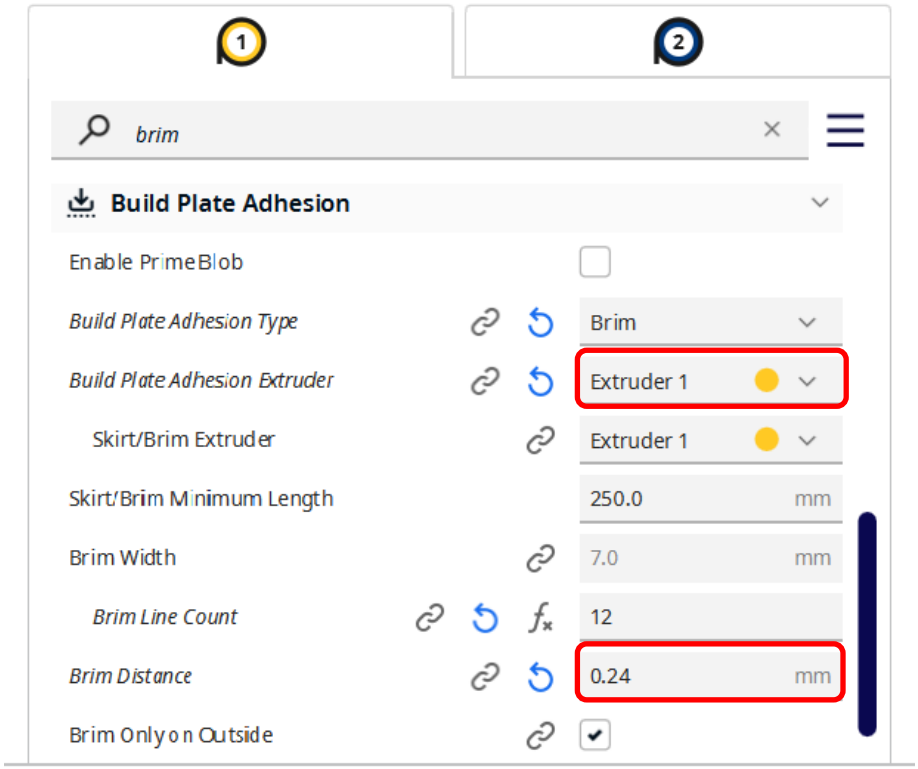
From experimentation with a specific printer, one can determine if a Brim is necessary.

This result shows that most of the Brim is printed using Extruder 1. The following page will show settings to change the Extruder that prints the Brim.

- at the top right of the Cura window, click on the **Settings** arrow
- select the **Extruder 1** tab
- click on the **Settings Options** icon and select **All** in the menu that appears




- in the **Search** box enter **brim** and scroll to the Build Plate Adhesion section
- change the **Build Plate Adhesion Extruder** to **Extruder 1**
- change **Brim Distance** to **0.24**




Changing the Brim Distance will allow the Brim to be more easily removed from the print, however, a value that is too large can make the Brim less effective.

The Brim Width and Brim Line Count can be different. These values may be reduced to reduce the Brim material.

# Using Cura – Single Component

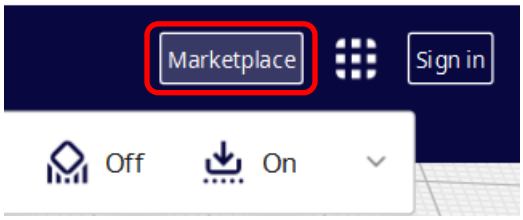
The previous section was for a model that was separated into 2 Components and thus 2 STL files. This section presents instructions for a single Component model and thus 1 STL file.

Because only a single STL file is imported, one cannot easily assign an extruder to different components. However, Cura does allow one to define regions of a model, that can be assigned different extruders. This method will use the **Support Blocker** tool. This tool is meant to define areas of a model where supports will not be added. We will not being using **supports**, but will use this tool to define regions to specify the Extruder to use for each region.

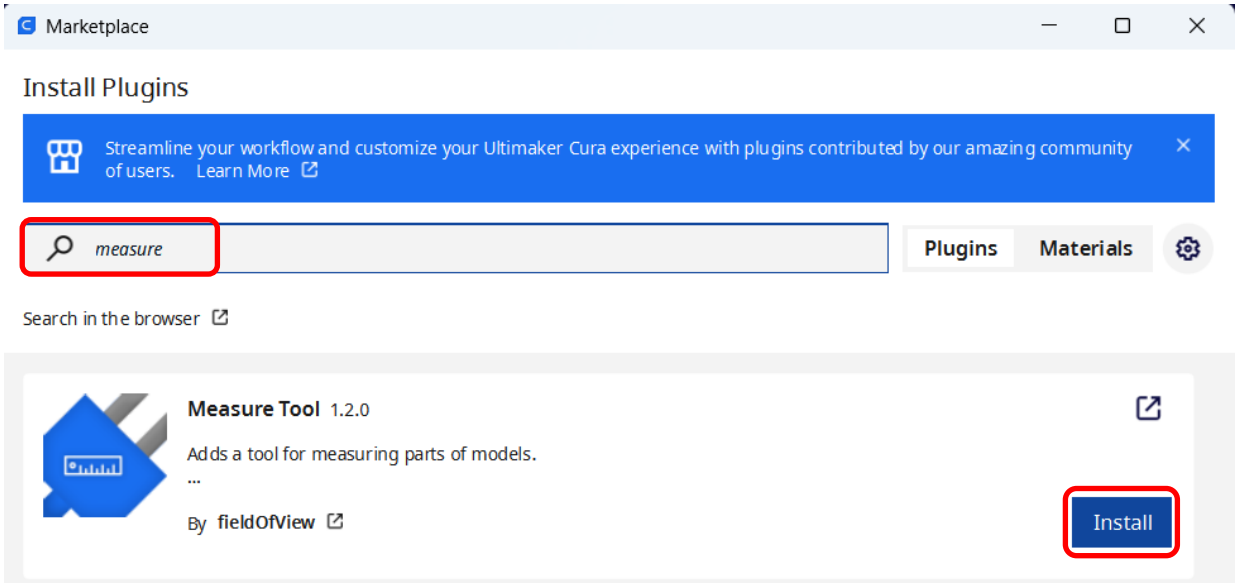
This method can be made easier by installing the Measure tool plug-in. One would think that Cura would have this built-in by default, but the feature has to be downloaded.

One should be able to download this tool without having to sign in or creating a Ultimaker account. If there are any issues with the download, the download of the measuring tool can be skipped.

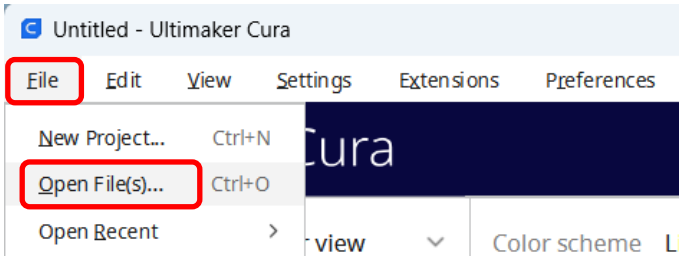
- at the top right of Cura, click on the **Marketplace** button



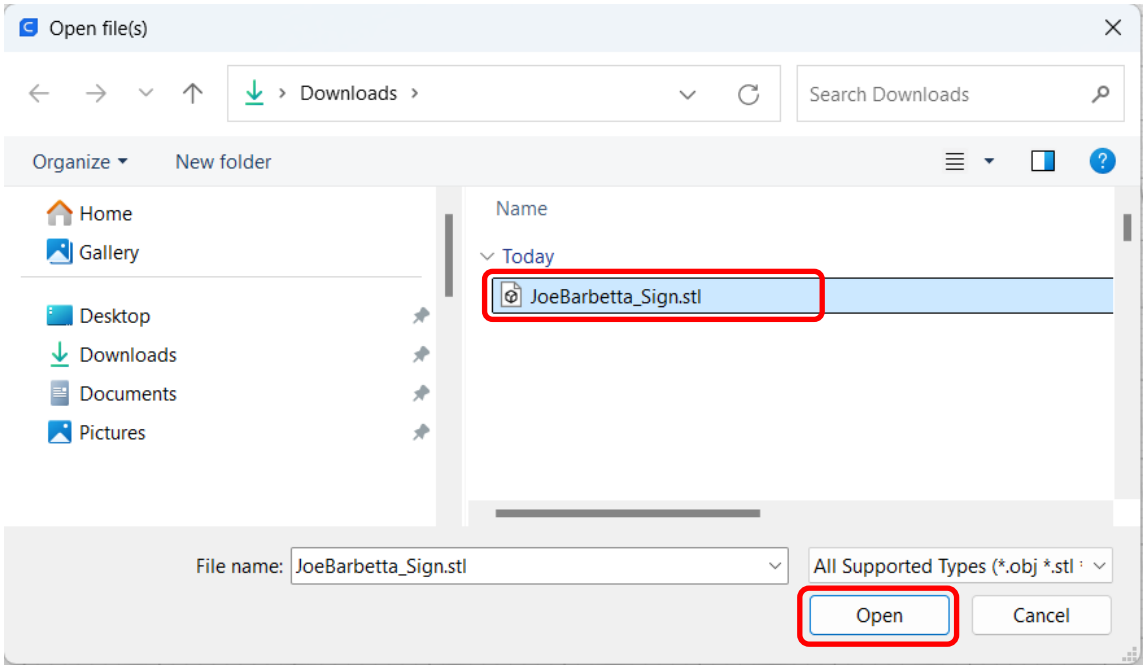

- type **measure** in the **Search box**
- click **Install** for the **Measure Tool** and then **Accept** for the License Agreement
- **Close** and **Reopen** Cura



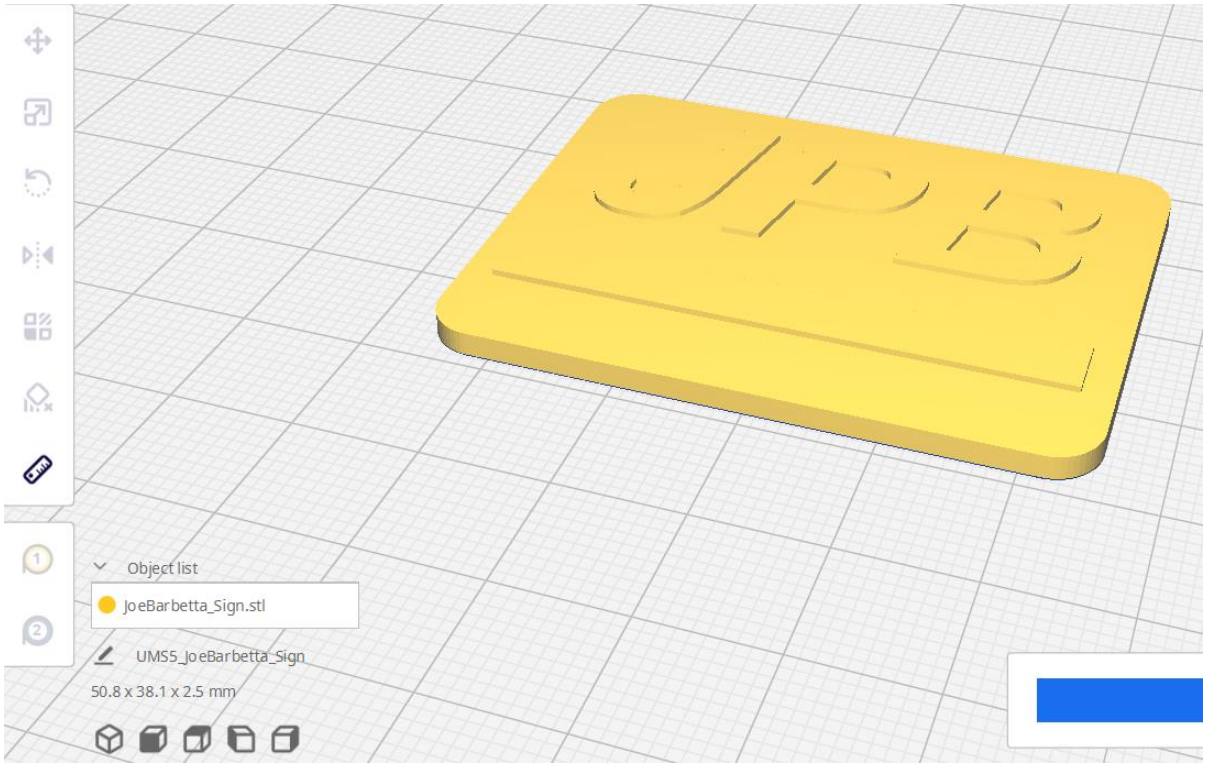
- from the **File** menu, select **Open File(s)…**



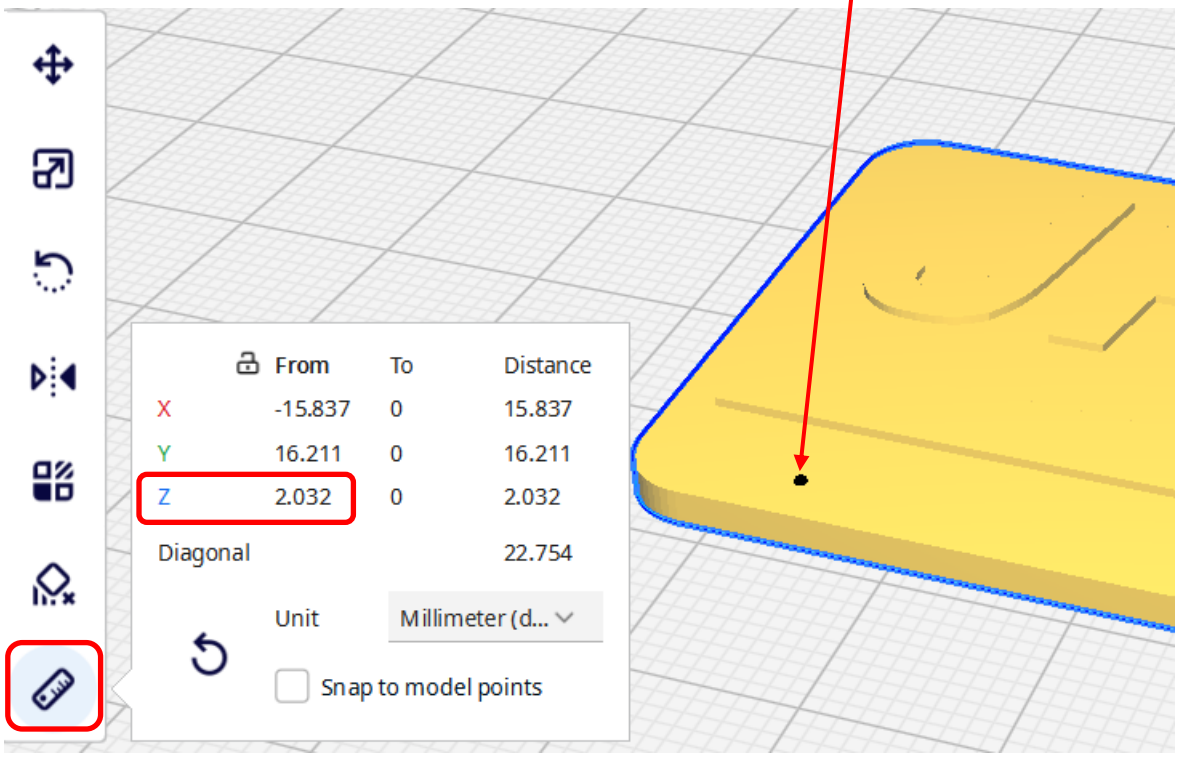

- navigate to the location of the files, which should be the **Downloads** folder, **select the file**, and click **Open**



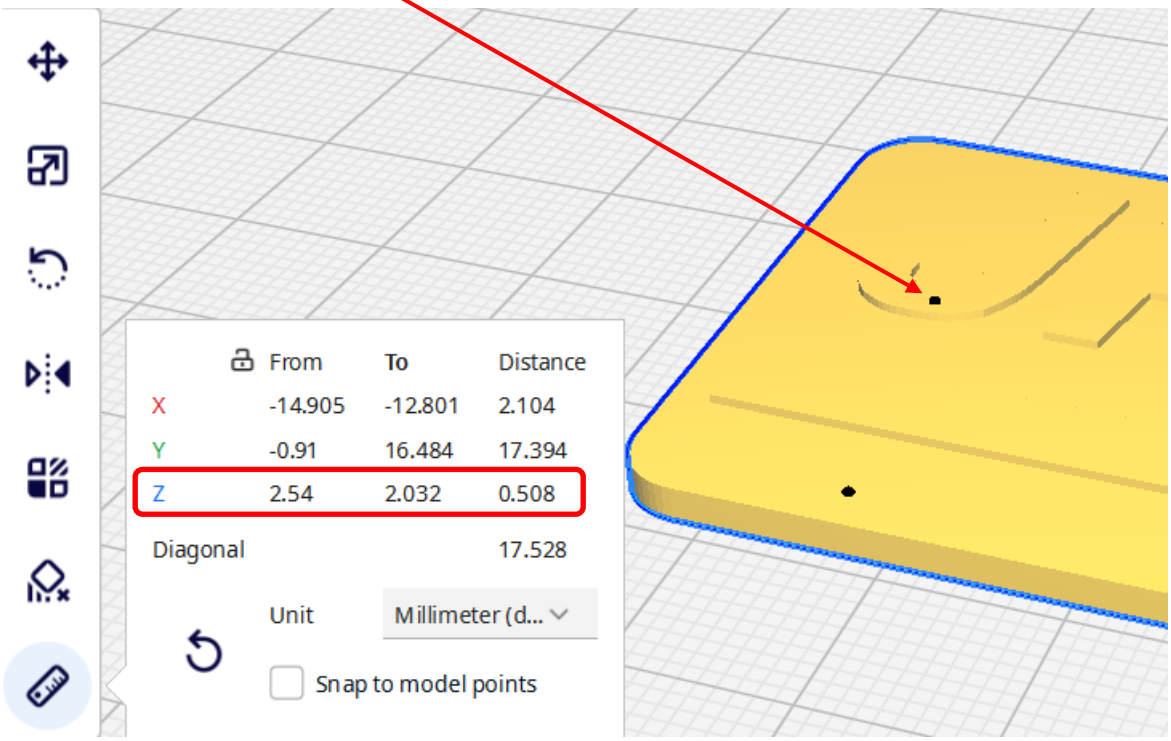

- use the **mouse wheel** to zoom into the model. Moving the mouse while **holding the mouse wheel down** allows panning the view. Right-clicking on the model or build plate allows changing the angle.



- if the **Measure** tool was installed, click on it an then click on the **top surface where there is no text or shapes** and make note of the **Z value**.  Here it is **2.032 mm**, which matches the 0.08 in that was used in Fusion.

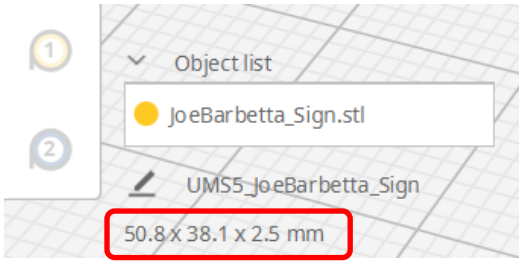



- click on the **top surface of a letter or shape** to add a 2nd point.  Here we can see that that Z value is 2.54 mm (0.100 in), which is the result of extruding the text and shapes 0.02 in above the 0.08 in substrate.  The Z Distance value of 0.508 mm matches the 0.02 in.

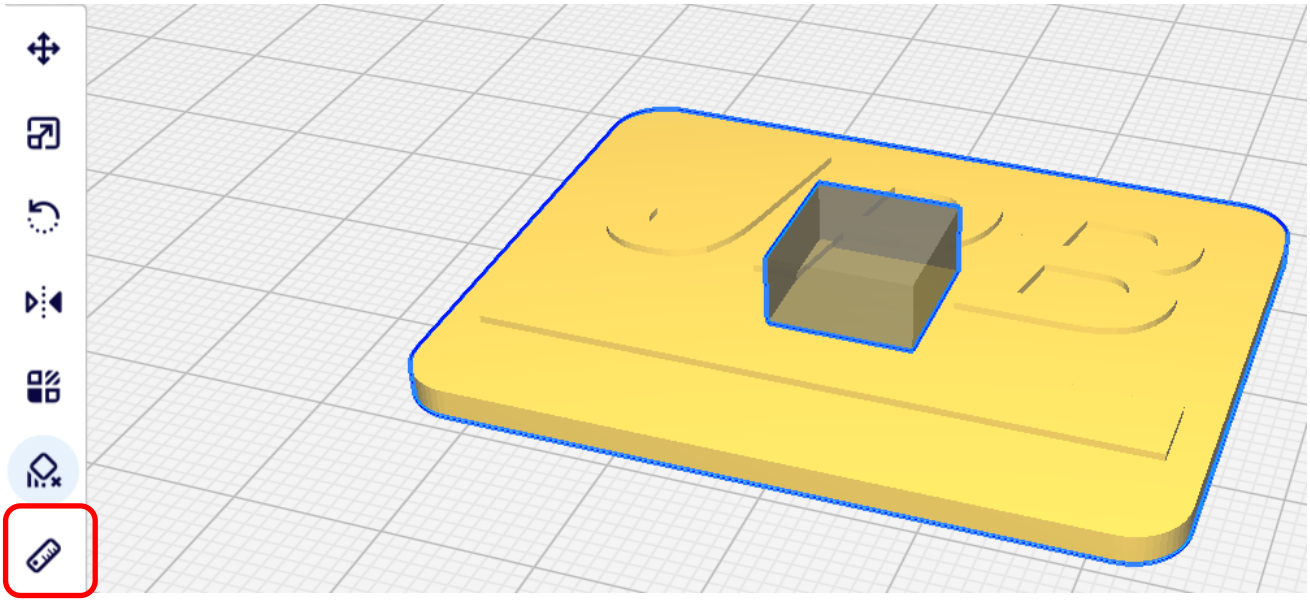


- note the **Dimensions** at the bottom left of the Cura screen. Here the Dimensions shows as **50.8 mm x 38.1 mm x 2.5 mm**, which corresponds to 2 in x 1.5 in x 0.1 in

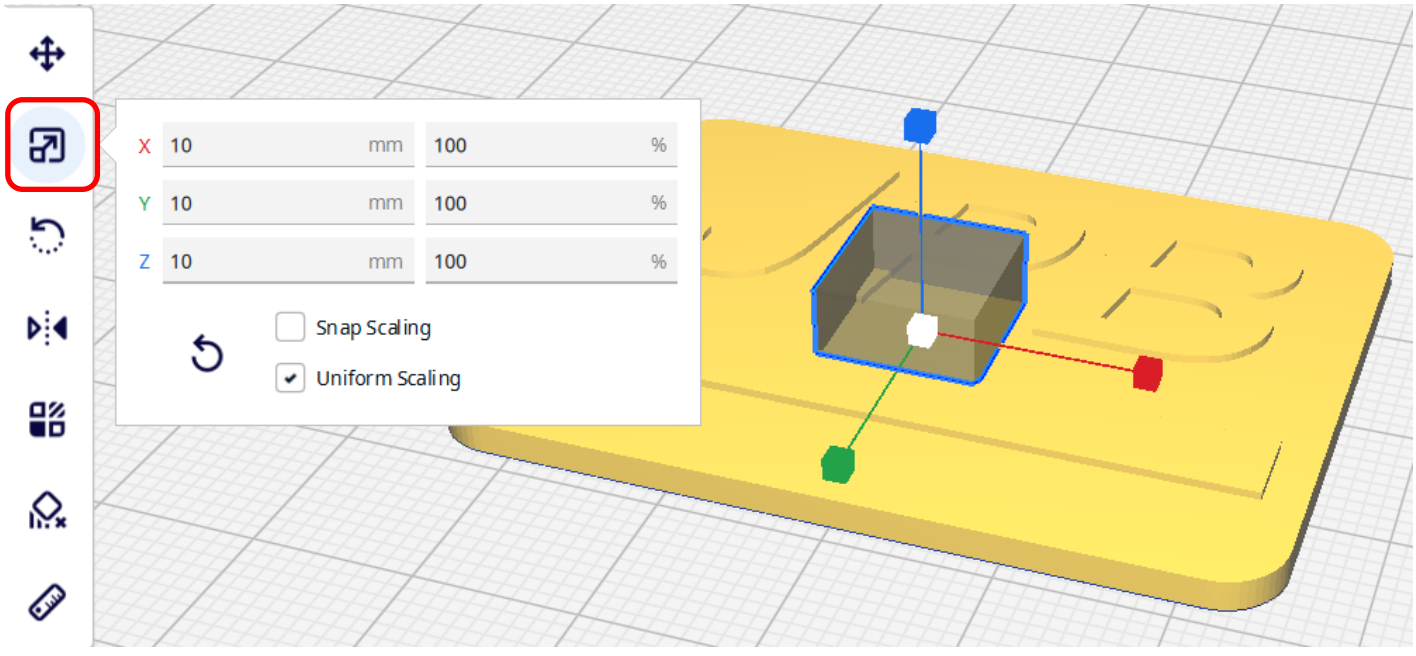
- add 4 mm to the Length and Width values, which results in **54.8 mm** and **42.1 mm**. These 2 values will be used to define the **Length and Width of the Support Blocker** that will be used for the substrate. The Height of the substrate, 2.032 mm, which was determined by the Measure tool, will be used for the **Height of the Support Blocker**. If the Measure tool was not available, we would know this value from that used in Fusion. 0.08 in = 2.032 mm



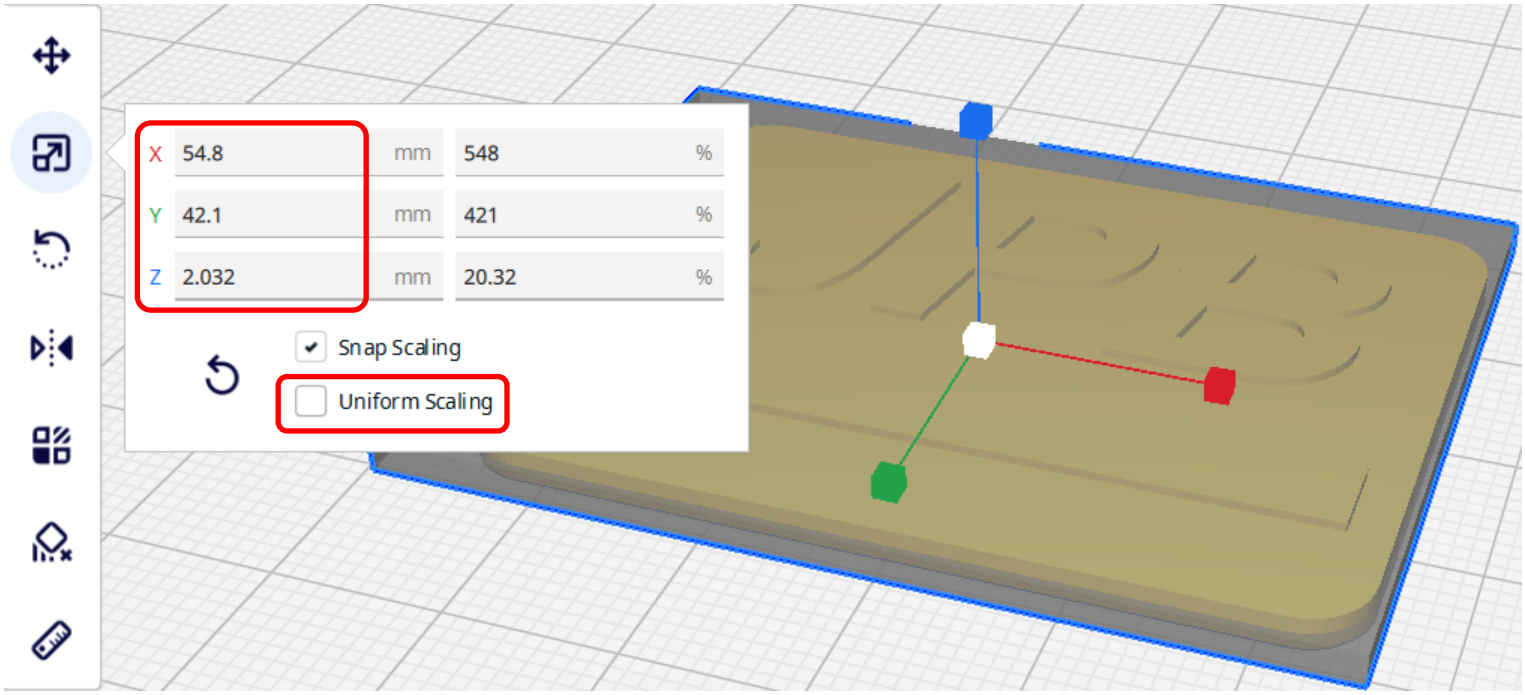

- click on the **Support Blocker** tool and then on the center of the model



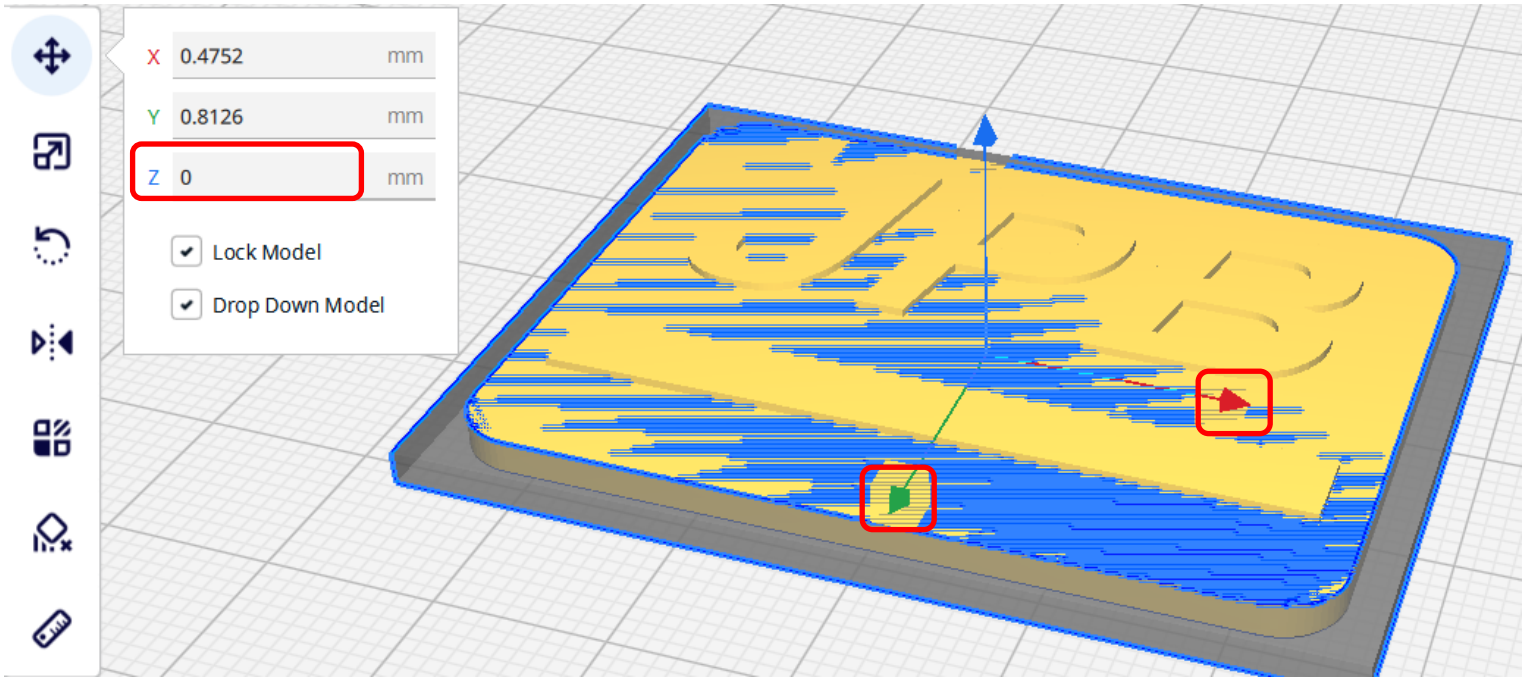
- select the **Scale** tool, which should show the initial dimensions



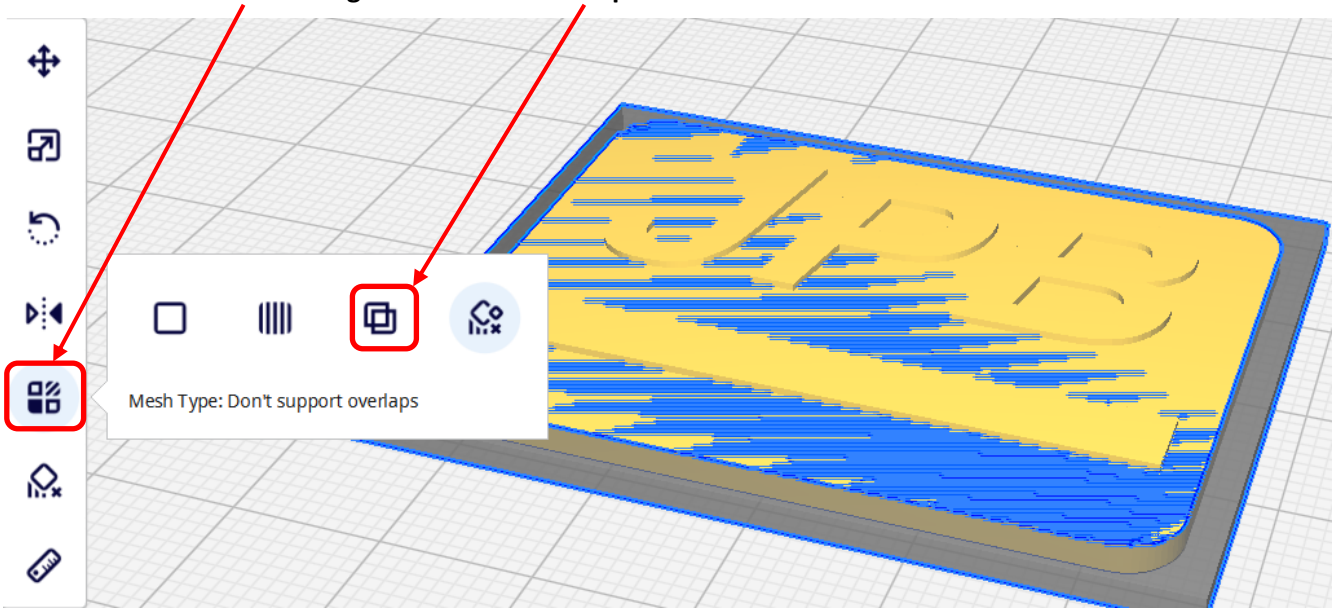
- uncheck **Uniform Scaling**
- set **X, Y, Z** to the **Length, Width, and Height** values, specified on the previous page



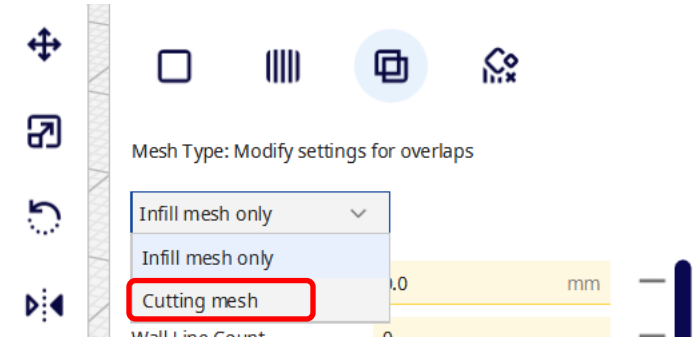

- select the **Move** tool use the **red and green arrows** to position the Support Blocker so that it surrounds the model. The centering is not critical. There just needs to be some Support Blocker material around each edge of the model.
- set **Z** to **0**



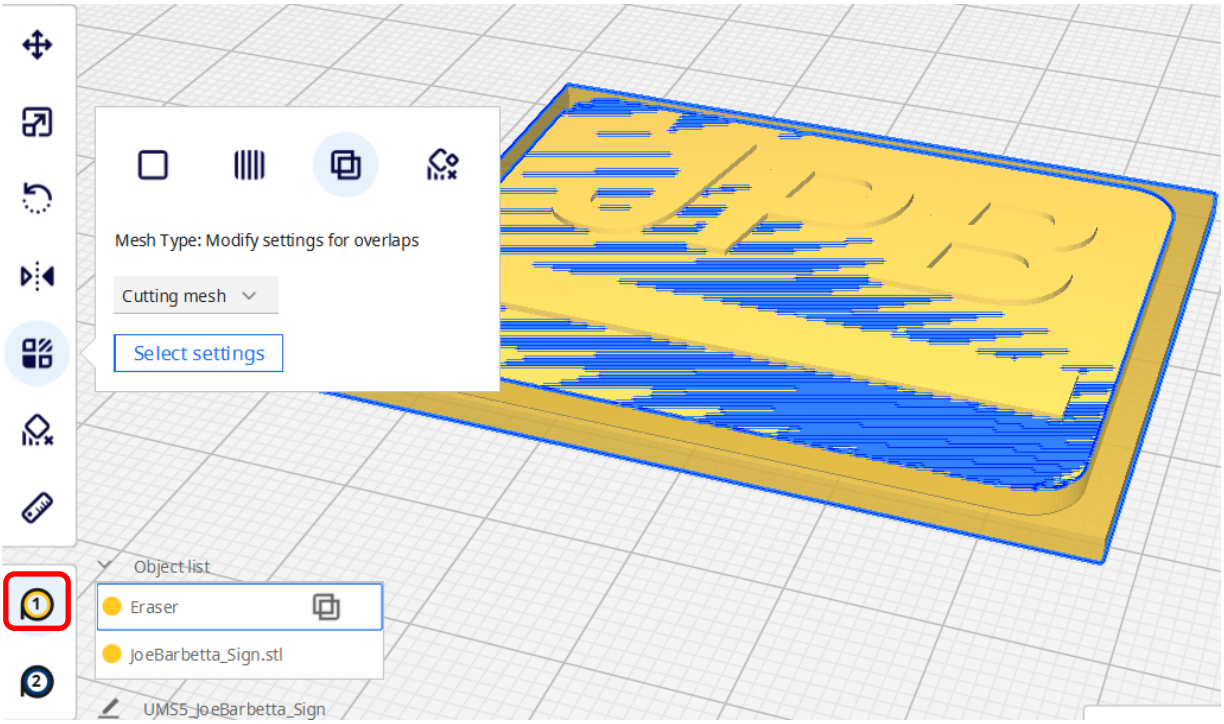
- select the **Per Model Settings** tool and the **Overlaps** icon




- change the setting to **Cutting mesh**

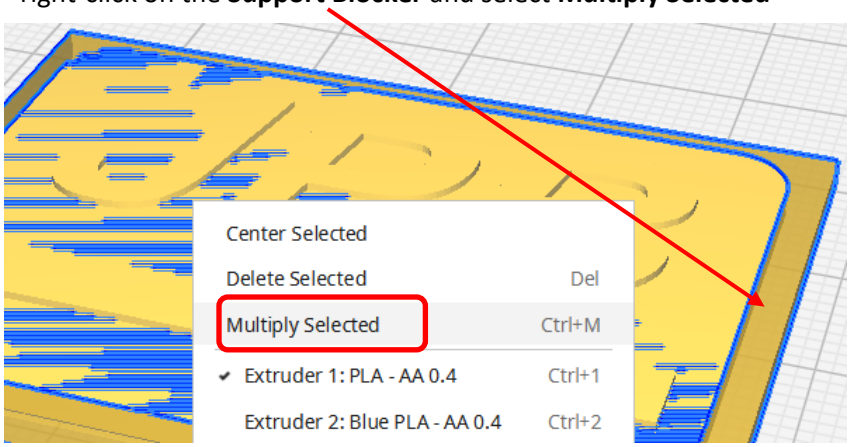



- click on the **Extruder 1** icon

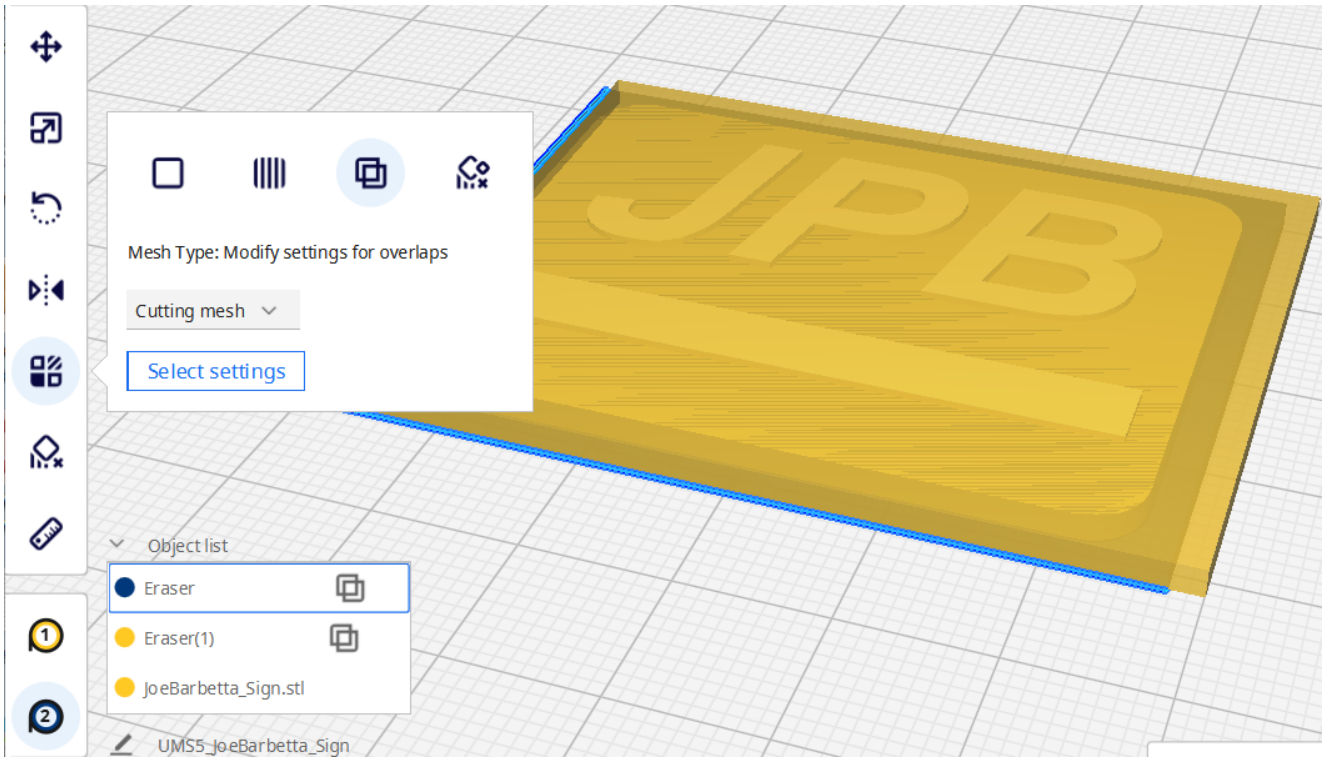


The following steps will create a 2nd Support Blocker that will be placed above the Support Blocker for the substrate and will thus contain the text and shape(s).

- right-click on the **Support Blocker** and select **Multiply Selected**

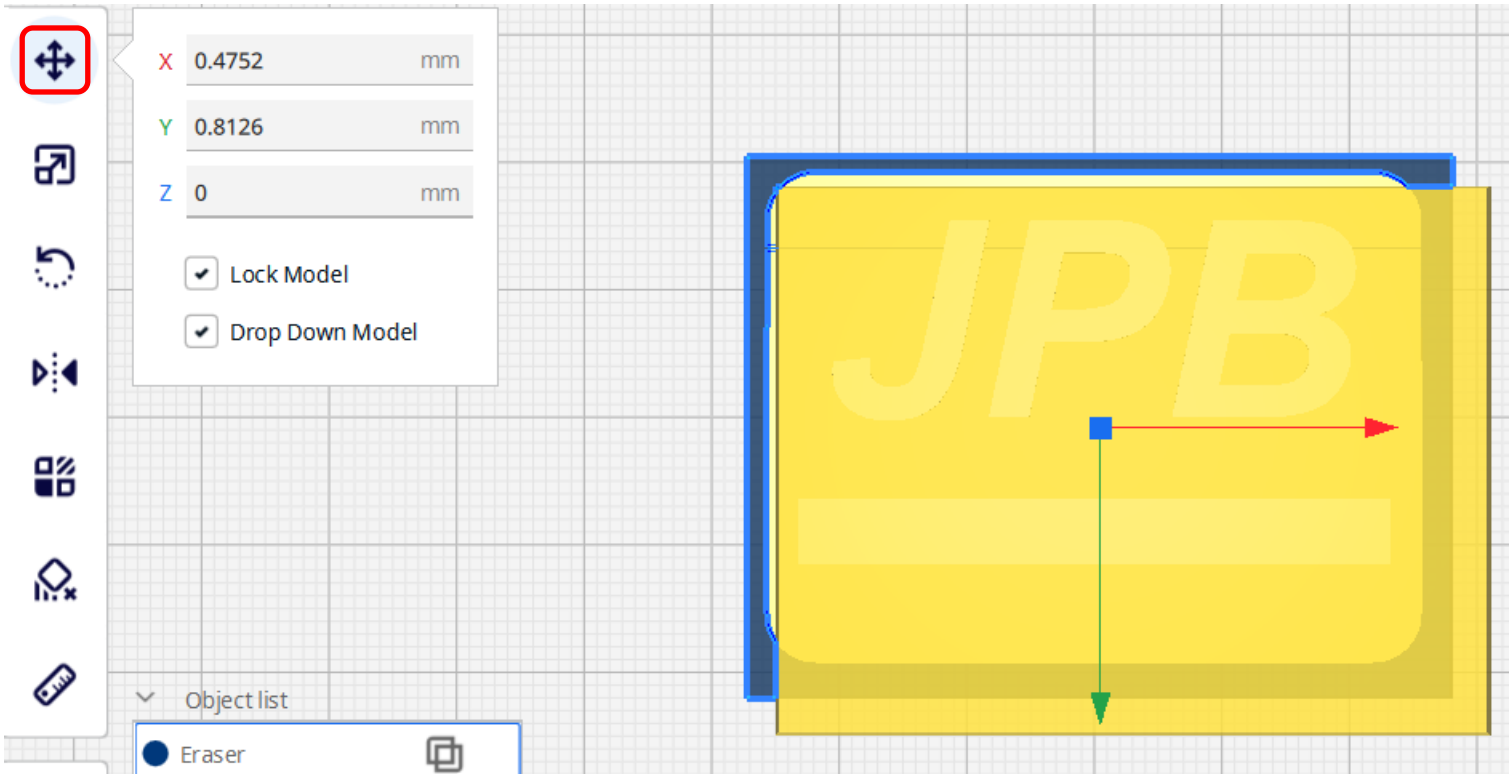



- click **OK**

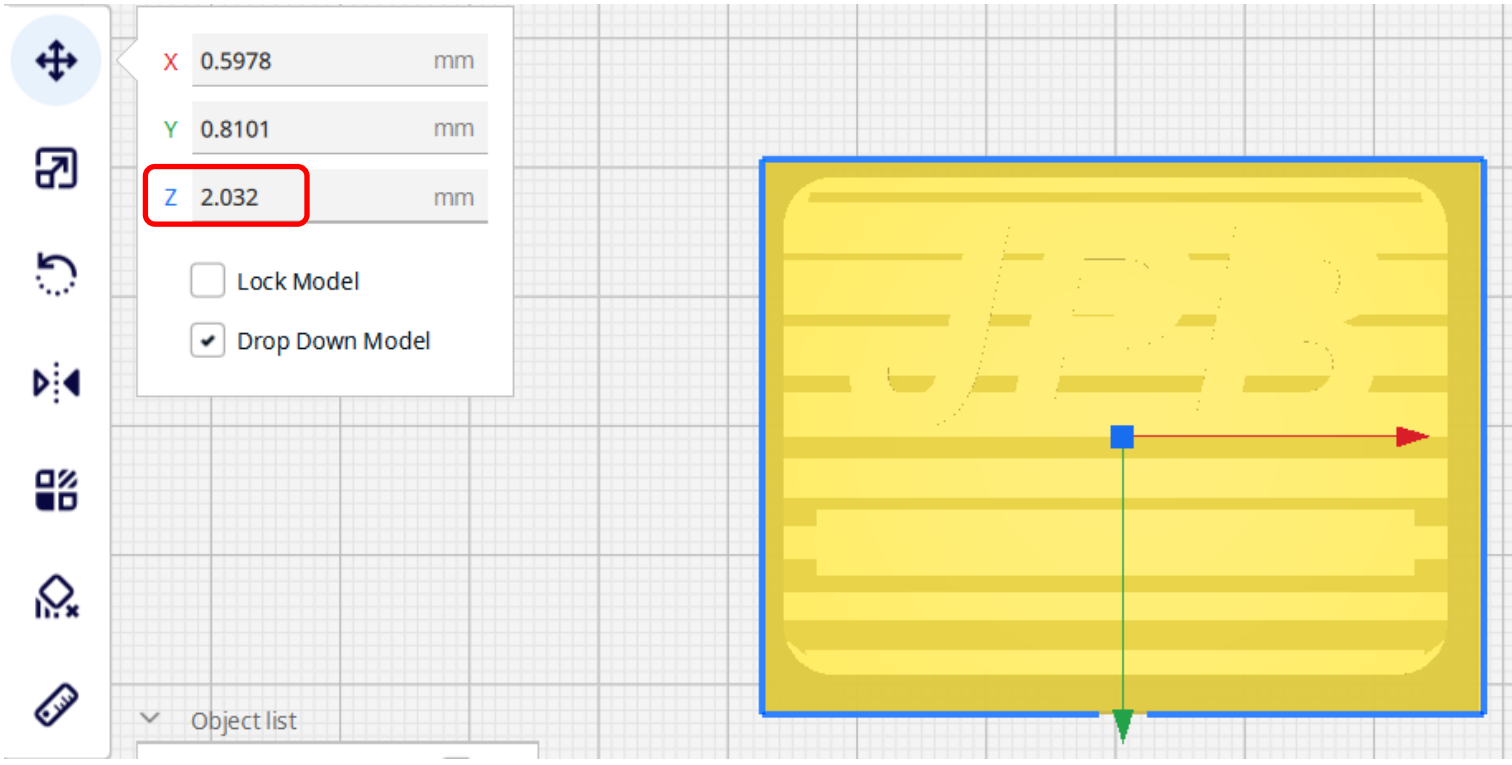



- select the **Top View** at the bottom left of the Cura screen

- select the Move tool and use the arrows to position the Support Blocker over the bottom one

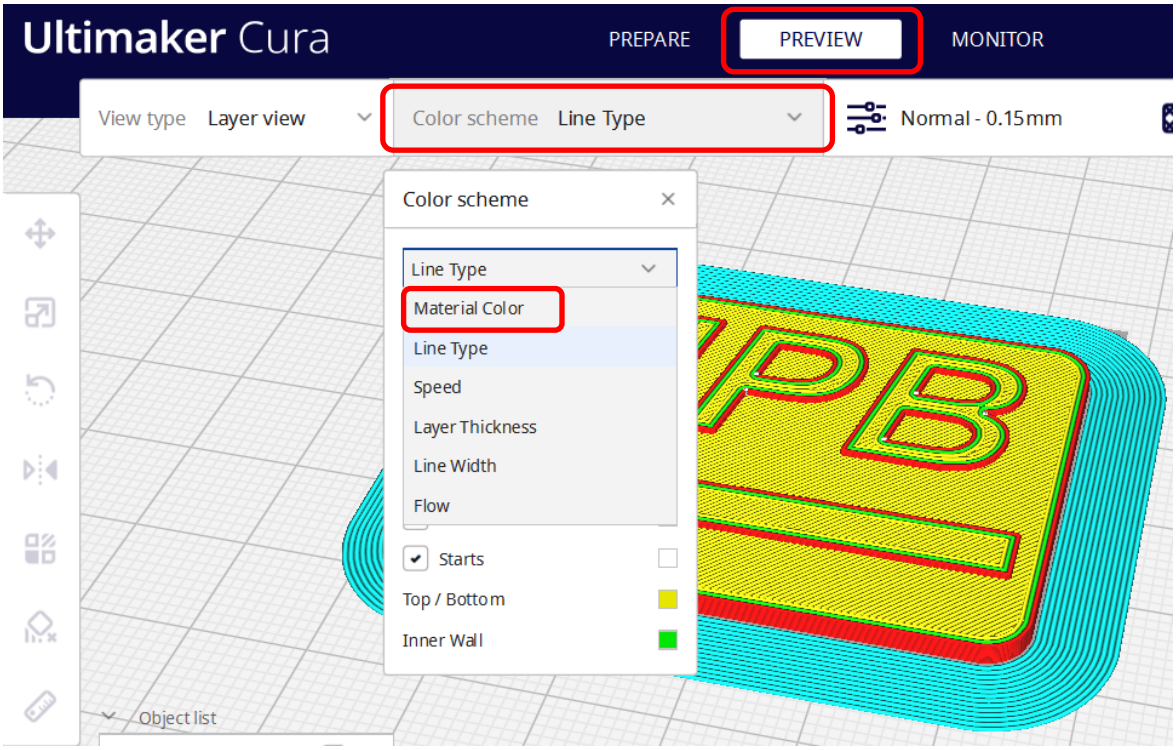



- set the **Z value** to **2.032**, which is the Height of the Support Blocker for the substrate

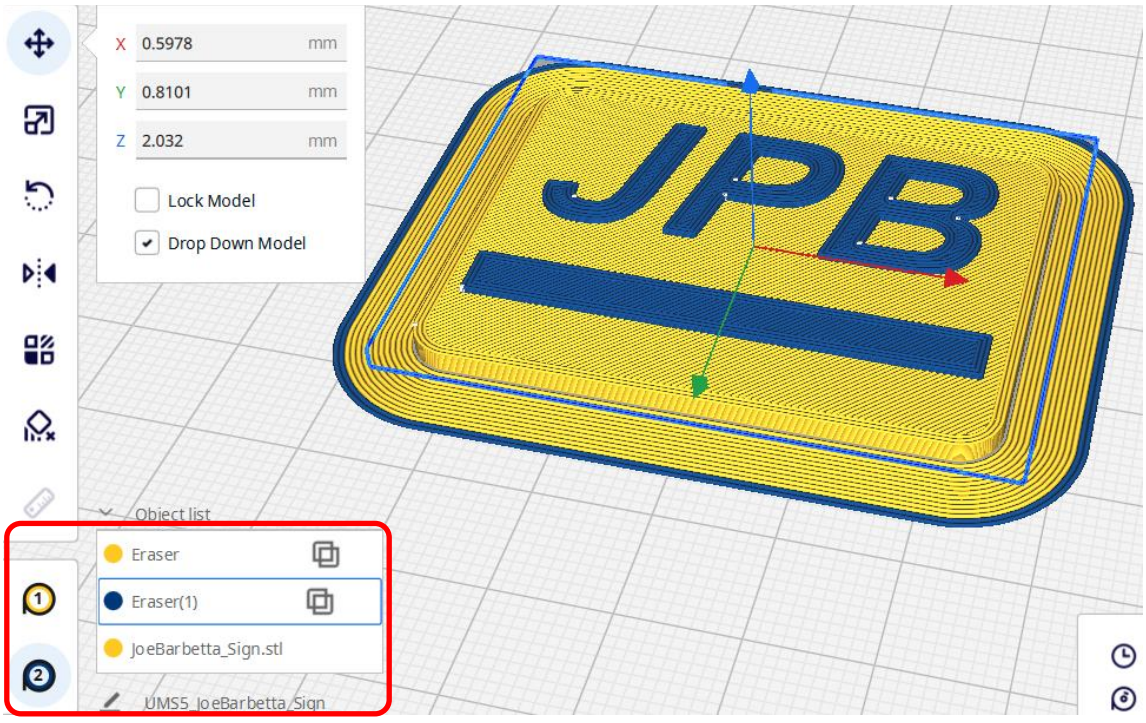


- click the bottom right **Slice** button


- click on the top center **PREVIEW** selection and then on the **Color scheme** options arrow
- change the **Color scheme** to **Material Color**




Note that the color is assigned according to Extruder number. If needed an Eraser can be selected and then an Extruder icon can be selected to reassign a Extruder and thus change the color. The top Eraser is for the Substrate and the bottom Eraser is for the Text and Shape(s). The Slice button must be clicked again.



# Using a Single Extruder Printer

This method will actually be the easiest to setup in Cura.

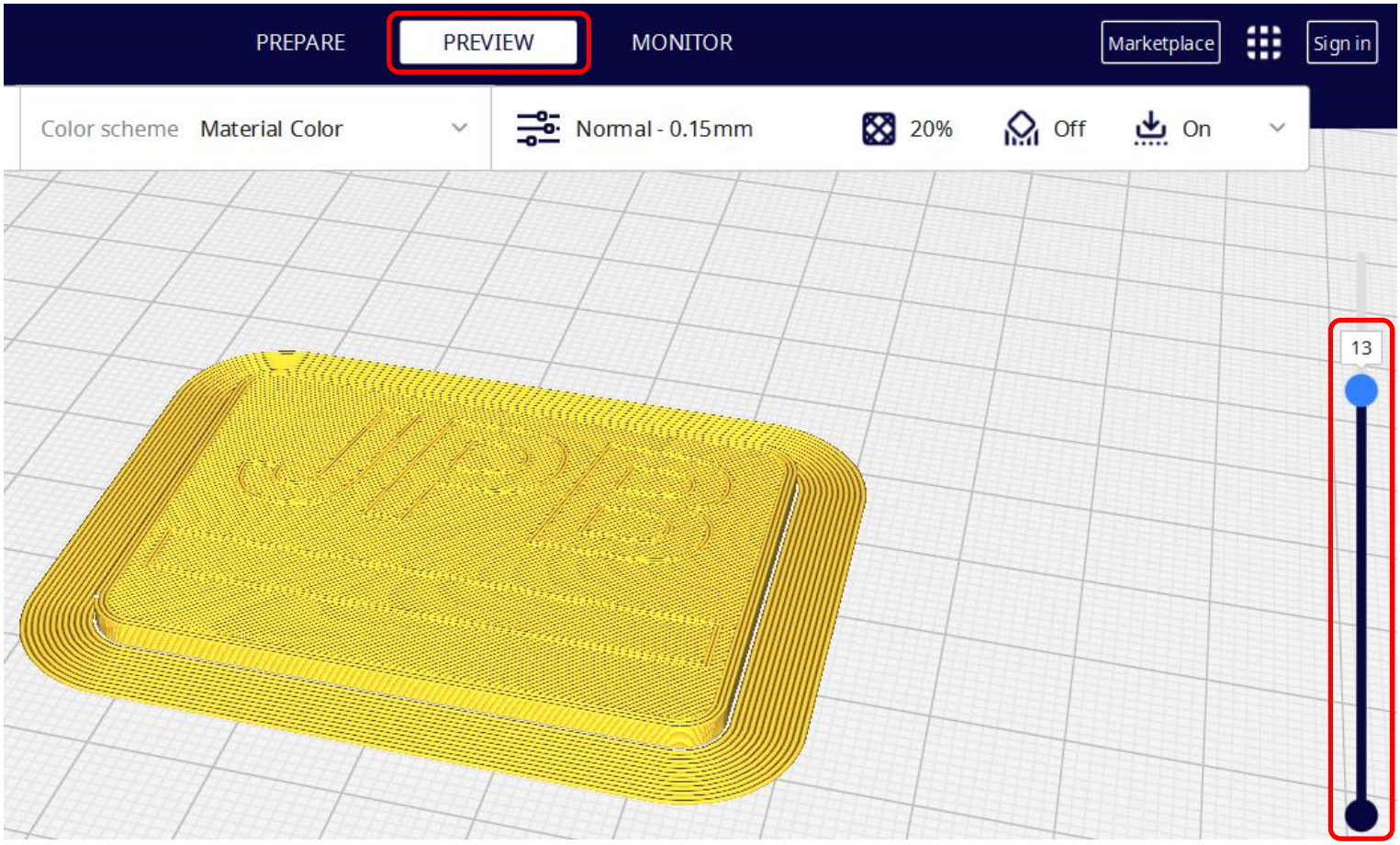
If the model is comprised of **2 Components**, follow the **Using Cura – Dual Component** section up to and including the point where **Merge Models** is used.

If the model is comprised of **1 Component**, follow the **Using Cura – Single Component** section for opening the file. The Measure tool is not needed.

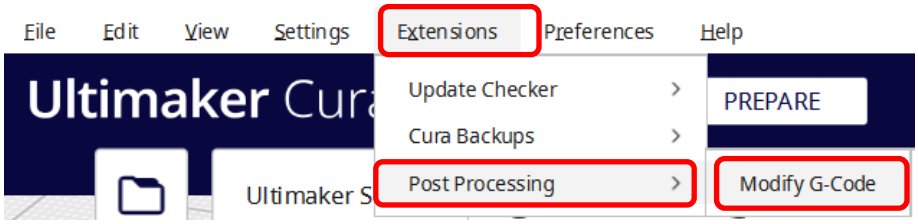
Once the model is open, one wants to create a pause after the layers for the Substrate are complete to allow one to manually change the filament. This can be straight forward in some slicer programs, but in Cura, at the time of writing, the G-Code must be modified.

- perform the Slice operation and then select Preview

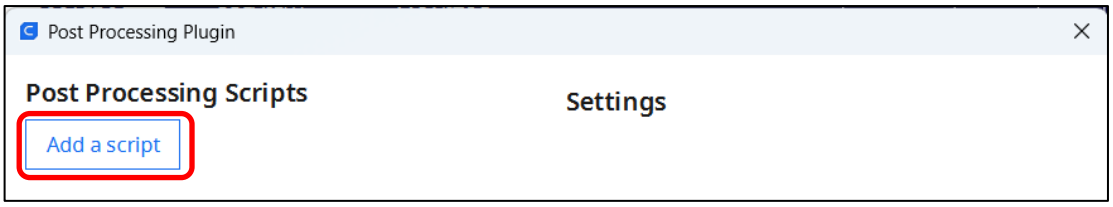
- move the right side slider to its bottom position and then move the slider up until the Text and Shape(s) become visible. Here that happened at layer 13. This determines that the last layer of the Substrate is 12.



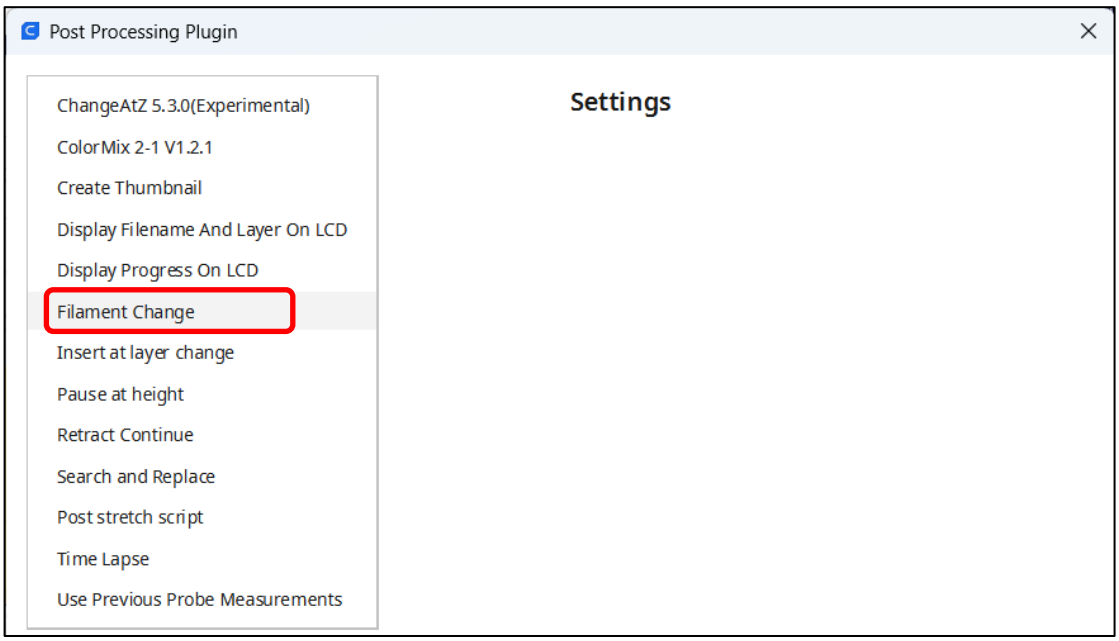
- from the **Extensions** menu, select **Post Processing** and **Modify G-Code**



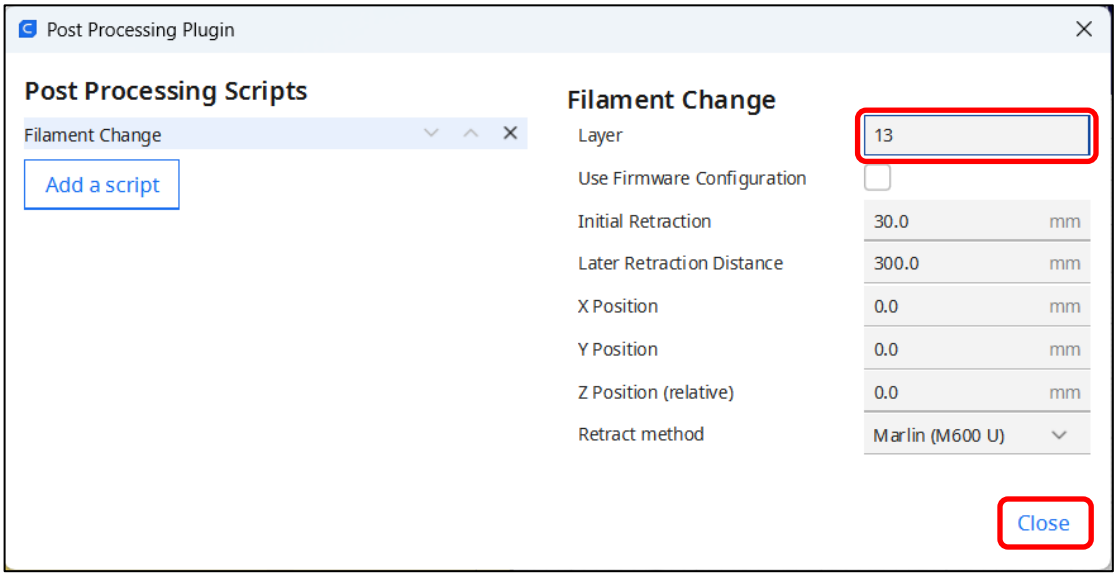

- click **Add a script**




- select **Filament Change**




- set the Layer value to the 1st layer of Text and Shape(s) and click **Close**




Research and experimentation should be done for the particular printer used.